CADprofi®

User's manual

We make effort to ensure that information contained in this user manual is accurate and up to date. However, if you see any mistakes – please let us know, we appreciate any suggestions.
We reserve the right to make changes in the program. Therefore, there may be slight discrepancies between this manual and the user program version.

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Introduction

About the manual

This manual describes the full functionality of **CADprofi®** application. We can find in it description of all commands and dialog windows as well as many useful tips and comments about using the application. We have also included the most important information required to properly install and run the program.

About the CADprofi program

CADprofi® is a comprehensive CAD application that vastly accelerates the design work in many branches. **CADprofi® Suite** program consists of the following modules:

- **CADprofi® HVAC and Piping**
- **CADprofi® Electrical**
- **CADprofi® Architectural**
- **CADprofi® Mechanical**
- **CP–Symbols** program contains a library of symbols from various branches as well as basic commands that allows editing schemes. The following libraries are available „HVAC & Piping", „Electrical", „Mechanical" and „Architectural".
- **CP–Manufacturers** program serves as an electronic CAD catalogs that contains libraries of world known manufacturers. **CP-Manufacturers** program is being distributed by manufacturers.

> ***Hint***
>
> ***CP-Symbols libraries and CP-Manufacturers catalogs are part of CADprofi® program.***

CADprofi program works as an "add-on" on various **CAD** programs. Therefore, in all descriptions the "CAD program" should be understood as the AutoCAD®, BricsCAD®, GstarCAD, IntelliCAD®, progeCAD®, ZWCAD etc. Most commands in these programs work identical.

> ***Notice***
>
> ***In AutoCAD® LT, you need to install additional software, such as LTX ™***

Important information about working with CADprofi

Working in **CADprofi** complies with general rules of designing in CAD programs. The most important benefit of using **CADprofi** program is the automation of most activities that are related to drawing and the comprehensive database of objects and solutions that are ready to use in your projects.

CADprofi commands have been prepared in such a way that user can work simple and intuitive even if he hasn't read the manual. However, we encourage reading the manual, so you will be able to fully use the program's potential.

In order to work comfortably in CADprofi application we recommend familiarizing yourself with the main program assumptions.

Compatibility with dwg, dxf formats

Project created in **CADprofi** are ordinary drawings that are saved in dwg or dxf formats. These drawings can be later edited in CAD programs even without loading the **CADprofi** add-on. Many of **CADprofi** objects are simple blocks that can be exploded and edited using CAD program standard commands. However, it must be noted that after exploding objects some properties may be lost and as a result it won't be possible to perform any commands or create any specifications on these exploded objects.

Standard drawing

For all new drawings, **CADprofi** inserts definition of layers, text styles, dimension styles and types of lines saved in a template file. For the appropriate CAD program the template file is the *normal.dwg* or *normal_ic.dwg* files, which are located in the *C:\Cadprofi\X.x\Block\General* folder. By editing the template file user can change or expand the standard definitions of loaded styles.

Notice

C:\CADprofi is a default installation folder of CADprofi program. This folder may differ if you have selected a different folder during the installation process.

Drawing scale

All projects should be carried out in a model space with a 1:1 scale in a chosen design unit (mm, cm, m, and inches).

The target scale specified by user should be determined during the printing or during the composing of sheets (in the paper space).

In some commands it is necessary to provide the scale. Usually this scale is only used to determine the correct text size for printing in a specified scale.

Linetype scale (line density)

It is possible to design in **CADprofi** in different units (mm, cm, m, and inches). In order to adjust the line density such as the density of a dashed line, **CADprofi** will automatically change the global line density ratio.

The default line density scale is 100 when designing in mm, 10 in cm and 0.1 in m. The default linetype scale is appropriate for construction and installation projects. In case of mechanical projects it is often needed to change the linetype scale.

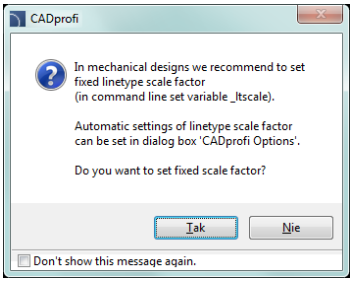
In order to change the linetype scale use the _LTSCALE command. Once the command has been started, user should enter the required value.

Example of linetype scale change:

```
Command: ltscale
New current value for LTSCALE <100>: 5
```

When you run certain commands (in the mechanical module), **CADprofi** will suggest to turn off the automatic adjustment of linetype scale.

Automatic adjustment of linetype scale can be turn on again at any time in the **CADprofi Options** dialog window.


CADprofi
In mechanical designs we recommend to set fixed linetype scale factor (in command line set variable _ltscale).
Automatic settings of linetype scale factor can be set in dialog box 'CADprofi Options'.
Do you want to set fixed scale factor?
Tak
Nie
Don't show this message again.

Layers

In **CADprofi Options** command user can turn off the automatic management of layers. However, it is not recommended because some layers properties are dependent on the objects. For example, when creating BOMs it is possible to automatically select objects from a particular module (for example all electrical symbols). If the automatic management of layers option is turned off then it won't be possible to identify symbols of a specific module.

Plot styles that specify the optimal line thickness on the printout are added to **CADprofi**. These styles match the automatic layers structure.

Covering objects (WIPEOUT)

Many **CADprofi** objects contain covering elements (_WIPEOUT). These elements cover elements lying underneath. To determine the display order, the CAD program command: _DRAWORDER (**Draw order**) should be used.

To correctly print drawings that contain these covered elements, user should properly set printing options or give a specific color to the CP_WIPEOUT layer.

Standard covers are being displayed without frames. Sometimes however, it may be necessary to display frames (e.g. to remove or edit the cover). To turn on the frames, it is possible to use the appropriate option in the _WIPEOUT command.

The coordinate system

Most **CADprofi** commands can be used in the user coordinate system (UCS). However, some commands require user to enable the world coordinate system (WCS), which is the default system in CAD programs.

No dialog windows (for example in the save command)

Some **CADprofi** commands block the standard dialog windows of CAD programs. After normal completion of such commands, **CADprofi** will restore the displaying of dialog windows. Sometimes user may stop the **CADprofi** command for example by pressing the **Esc** key. In such cases, **CADprofi** may not restore the dialog windows.

In order to restore the displaying of standard dialog windows user should set the FILEDIA and CMDDIA variables to 1. To do this, user has to type in the command line the variable name and set its value to 1.

```
Command: filedia
New current value for FILEDIA <0>: 1
Command: cmddia
New current value for CMDDIA <0>: 1
```

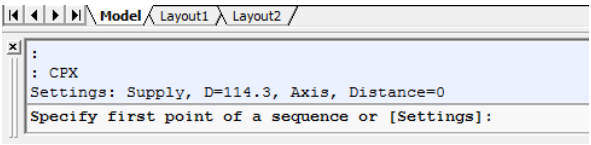
User can restore displaying of dialog windows by using the **CADprofi Options** command.

Toolbars and tool palettes

In CAD programs it is possible to adapt menus and toolbars. Using the **CADprofi Configuration** program, **CADprofi** application can be “attached” or “detached” from the CAD program. When user attaches **CADprofi** application to the CAD program, **CADprofi** standard menu and toolbars will be loaded. In this case, any changes made to toolbars and menus will be overwritten. This also applies to tool palettes and **CADprofi** ribbons.

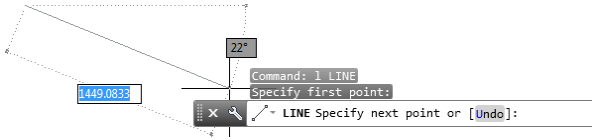
Command line

In some CAD programs such as new versions of AutoCAD®, user can disable the command line. Both CAD programs and **CADprofi** application display important information in the command line. For this reason, user should have the command line enabled with at least two visible lines.



Command line in AutoCAD® program

In AutoCAD® 2006 and later, user can enable the dynamic command line. Dynamic command line displays only one line and furthermore it may slow the **CADprofi** application (for example when editing multiple objects simultaneously). That's why the dynamic line is turned off by many **CADprofi** commands.



Dynamic command line in AutoCAD® 2006 program and later

Particularly inconvenient is turning off both classical and dynamic command line. In this case, user won't be informed about the state of currently processed command.

Expanding libraries and manufacturers' catalogs

It is possible to expand both symbols' and objects' libraries in **CADprofi**. Once the program has been installed, it is possible to install new manufacturers' libraries such as those that have been developed after purchasing **CADprofi**. Both users' and manufacturers' libraries are saved in the *Producers* folder. This folder is located in the main program folder (e.g. *C:\Cadprofi*). In order to save all the changes made in the manufacturers' libraries as well as in the user symbols' and objects' libraries, the *Producers* folder with all or with selected libraries should be backed up (archived).

After a program reinstallation or transferring to another computer user can transfer all the content from the *Producers* folder to a new installation. It is also possible to copy libraries within one team of designers (in one company).

User database - UserData folder

In **CADprofi** program standard databases have been separated from the user databases. Thanks to this it is possible to update the program without losing the data stored by the user.

All user data is stored in the *UserData* folder. This folder is located in the main program folder (e.g. *C:**Cadprofi*). In order to save all changes in the user databases, user should archive the entire *UserData* folder. After a program reinstallation or transferring to another computer user can transfer all the content from the *UserData* folder to a new installation. It is also possible to copy *UserData* within one team of designers (for example in a single company).

In **CADprofi** program user can define various parameters or expand standard databases. It is possible, for example, to define additional types of installations, additional types of cables and additional objects' parameters (e.g. price, currency, color, etc.).
This data is stored in the *UserData* folder.

Installation

The process of creating a **CADprofi** seat consists of 3 steps: installation, configuration with your CAD program and license activation.

> ***Notice***
>
> ***In order to correctly handle the installation process and program configuration user needs the administrator or power user permission.***
>
> ***CADprofi program allows creating your own libraries, adding your own types of cables etc. This data is stored in different folders that's why users need to have rights to save and edit files in all folders in which the program is installed. For the same reason it is not recommended to install the program on the Windows Vista, Windows 7 or later operating systems in the "C: \ Program Files (x86)" and "C: \ Program Files" folders because in those folders the above operating systems are blocking the possibility to save and edit all files.***

Program installation

After placing the DVD in the disk drive a dialog window will appear allowing you to choose the language in which the installation will be done.

> ***Hint***
>
> ***If the start window won't appear automatically then run the autorun.exe file manually from the Start folder located on the CADprofi DVD.***

In order to start the installation wizard choose the **CADprofi – Installation** item and follow the prompts.

Selecting components

DVD disk contains all **CADprofi** modules. During the installation it is possible to install all **CADprofi** modules, yet the user must be aware that during the activation he will receive activation codes only for the purchased modules. Modules, which were not activated, will be blocked after 30 days from the first program run.

Online version

Installation file can be downloaded directly from the **CADprofi** server. This option is available when user has purchased the online version.
Installation file with all manufacturers' libraries has a size of over 1 GB. The name of the file is for example: „CADprofi_*x_full_dvd.exe* ".
If the user doesn't need all libraries he can download a smaller file e.g. „ *CADprofi_x.exe*".
After installing **CADprofi** from this file user will be able to download libraries of selected manufacturers through an online update system on a later date.
It is necessary to copy the downloaded files to any data storage device such as a DVD disk. In this way it will be possible to install the program after a computer replacement or a computer crash.
It is also recommended to archive both *Producer* and *UserData* folders in which symbols and data defined by user are located. Thanks to this, if needed, it will be possible to quickly restore the whole program content from the backup.

Archival versions of CADprofi program

Archival versions of **CADprofi** software are available on http://www.cadprofi.com website.
CADprofi programs are being covered by a time subscription, so when user downloads the archive version, he should download the correct one (the one that was published at the time when user's subscription was valid).

Notice

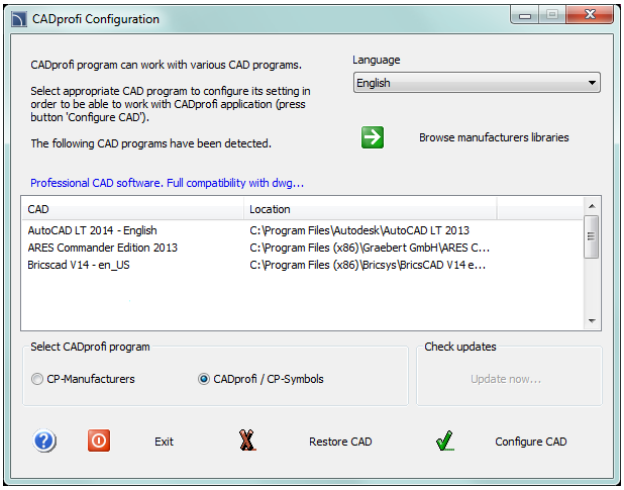
If user has downloaded version that had been published after the purchase date or after the subscription expiration date then he won't be able to activate the program.

Configuring CADprofi with the CAD program

Automatic configuration of CADprofi with the CAD program

Because **CADprofi** application works on many CAD programs it is necessary to make the configuration process. This process is being carried because it is necessary to assign the **CADprofi** application to the right environment, which will enable **CADprofi** to load automatically after user runs the CAD program.

In order to do this run the **CADprofi configuration** file, which shortcut has been placed on the desktop after the installation process.


CADprofi Configuration
CADprofi program can work with various CAD programs.
Select appropriate CAD program to configure its setting in order to be able to work with CADprofi application (press button 'Configure CAD').
The following CAD programs have been detected.
Language
English
Browse manufacturers libraries
Professional CAD software. Full compatibility with dwg...
CAD
Location
AutoCAD LT 2014 - English
C:\Program Files\Autodesk\AutoCAD LT 2013
ARES Commander Edition 2013
C:\Program Files (x86)\Graebert GmbH\ARES C...
Bricscad V14 - en_US
C:\Program Files (x86)\Bricsys\BricsCAD V14 e...
Select CADprofi program
CP-Manufacturers
CADprofi / CP-Symbols
Check updates
Update now...
Exit
Restore CAD
Configure CAD

CADprofi Configuration dialog window

Dialog window contains the following items:

Language – list of available languages in **CADprofi** software.

Browse manufacturers' libraries – allows checking the content of manufacturers' libraries.

Found CAD programs – list of all CAD programs installed on your computer, to which **CADprofi** can be automatically configured.

Select CADprofi program – choose which **CADprofi** application should be configured with the CAD program.

Check updates – check the availability of a new version via **Online update**.

Restore CAD – allows to „disconnect" **CADprofi** application from the selected CAD program (work without **CADprofi**).

Configure CAD – configure **CADprofi** application with the selected CAD program.

Procedures

Configuring CADprofi with the CAD program

1. Run the Click the **CADprofi** configuration program that is located on a Windows desktop. A new dialog window named **CADprofi Configuration** will appear.
2. Select the language that will be used in the **CADprofi** program.
3. Select the CAD program on which you want to use **CADprofi**.
4. Make sure that you have selected **CADprofi** program. (**CP–Manufacturers** and **CP–Symbols** are used to configure other programs).
5. Click the **Configure CAD** button in order to start working with **CADprofi**.
6. (Optional) Application may require to run it with higher rights, and therefore will require an administration inervention.
7. After a proper configuration a new dialog window will appear asking us if we want to run the CAD program.

> ***Notice***
>
> ***Configuring CADprofi with your CAD program is a single operation.***
> ***After that run the CAD program using its own shortcut.***

Manual configuration of CADprofi with the CAD program

If the CAD program that you are currently using is not displayed in the **CADprofi Configuration** dialog box, then you can perform a "manual" configuration. To make the process successful we need to be sure that we are using a CAD program that is compatible with **CADprofi**. Additional information can be found in the *IcadConf.utc* file, which is located in the main **CADprofi** folder. This file can be opened with any text editor.

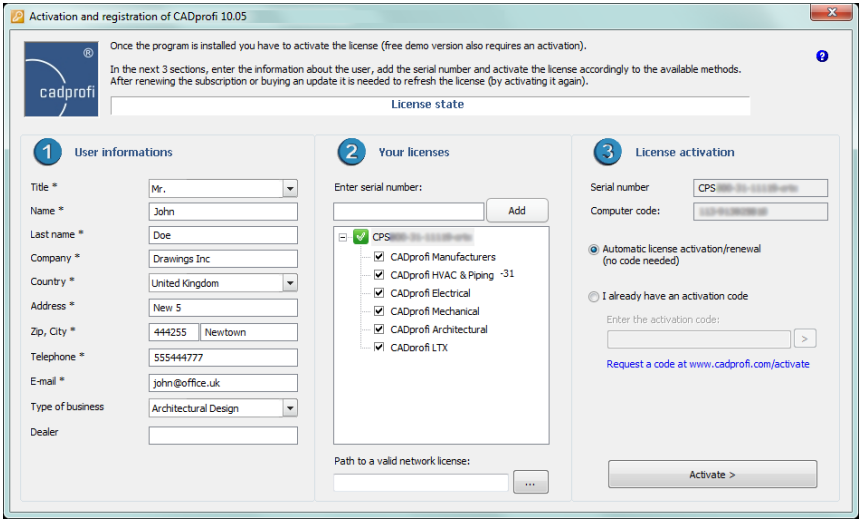
Procedures

Manual configuration of CADprofi with the CAD program

1. **Adding CADprofi folder to a support file search path.**
   To the list of support file search path in the CAD program add the location of main **CADprofi** folder such as *C:\Cadprofi\X.x*. In most CAD programs this list is supported by a SRCHPATH variable or with the **Options** command.
2. **Loading the right CADprofi menu file**.
   With the _MENULOAD or _CUILOAD command load the appropriate **CADprofi** *mnu* or *cui* file.
3. **Loading the application (the appropriate executable file) and adding it to the start list.**
   With the _APPLOAD command load the appropriate executable file (dll, brx, zrx, grx, arx or other). If we want it to work after each time we run the CAD program we should add the executable file to the start list.

Registering and activating the license

In order to activate the commercial version of **CADprofi** application you need to perform the process of activating the license. The **activation of license** window will be displayed each time the CAD program is started until the license is correctly activated.



Dialog window used for registration and license activation

Dialog window contains:

User data – data required for user registration.

Enter serial number – serial number of your license. This number can be found in the **CADprofi** package or in the email that will be sent after program purchase.

Path to a valid network license – field available in case of activating a network license (check page 21).

License activation – information and commands necessary to activate the license.

Computer code – your computer's unique code required to identify your license. This number is generated automatically.

Activate > – button that allows to automatically update or renew the license from the **CADprofi** server.

Automatic license activation/renewal – this option allows to activate the rights by connecting to the **CADprofi** license server.

I already have an activation code – this option allows to enter the activation code that was previously downloaded from the **CADprofi** website. This option will be unlocked after failed automatic activation attempt.

Request a code at: – a link to a website that is used to activate the **CADprofi** license.

Procedures

Single license activation

1. Run the CAD program, to which **CADprofi** has been configured and wait for the **Registration of application** window to appear. Registration window can also be displayed by clicking the **CADprofi - About** command.
2. Fill out user data.
3. Enter your **serial number** and click **Add**.
4. Click the **Activate >** button in order to automatically activate the license.
5. (Optional) Click the **Request a code at…** link in order to receive **an activation code** on **CADprofi** website. Afterwards click the **I already have an activation code** option thanks to which it will be possible to enter the code in the appropriate field.

Notice

If the customer possesses several serial numbers from CADprofi or CP-Symbols programs then the activation procedure must be repeated for each serial number.

Authorization codes are only valid for a single computer. If it's necessary to transfer the license to another computer user has to uninstall his program on the currently used computer and afterwards install the program on a new computer and activate the program again.

If you purchase the program update or you renew the subscription it is necessary to reactivate the license in order to refresh information about license on your computer.

Network license

Network license allows user to use the **CADprofi** software in a specific **LAN** network (**L**ocal **A**rea **N**etwork). With this license **CADprofi** can be installed on any number of computers that are used in a specified network, while the number of ordered licenses defines the number of users that can use the program simultaneously.

Information and technical conditions

After installing the program on one of the workstations, user has to activate the network license. As an outcome in a specified by the registering user folder a file with the network license will be created. This file divides permissions between users, according on the program startup order.

Notice:

**** Workstations, that use the license must have rights to save in the selected folder.***
**** On each workstation, the license path must be defined in an identical way. It is not possible to use letters of mapped drives.***
**** Examples of correct network pathes:***
- with server name: \\ServerName\Share\Folder
- with server IP address: \\192.168.1.17\Share\Folder
**** Network folder, that was specified during activation, can be later changed only in a request sent to the producer o the program (the licensor).***
**** At least one of the workstations must be connected with the internet in order to periodically, automatically refresh the data in the license file.***

Using the network licenses

Each workstation gets whole seat from the given network license. If a network license is for example for 5 seats and contains both **HVAC & Piping** and **Electrical** modules, then each workstation will get both modules, allowing to work 5 users simultaneously. In order to manage separately with each of the modules, user should order separate network licenses for each branch.

Procedures

Activating the network license

1. Run the CAD program, which was configured with **CADprofi** application and wait until the **Program Registration** window will be displayed. Registration window can also be displayed by clicking the **CADprofi - About** command.
2. Fill out user data in order to register the program.
3. Enter the **network license Serial Number** and click **Add**.
4. In the **Path to a valid network license**, specify the location, where the network license file will be created for the specified serial number.
5. Click the **Activate >** button in order to automatically obtain the **Activation code** and **register the user**. This activation is a once a time process and can be done on any computer that is in the network.
6. Enter the same **CADprofi network license Serial Number** on all work stations and provide the same **Path to a valid network license**. Wait till the application will download all the necessary permissions.

> ***Notice***
>
> ***The network license path just as the computer number is involved in activating the license, therefore it must be the same on all work stations that use the same license file. It is recommended to use the IP address in the path, instead than for example a mapped network drive.***

Returning the network license

1. In the program from the menu or the **CADprofi** toolbar select the **CADprofi - About** command and then open the **Program registration** window.
2. Click the right mouse button on the **CADprofi network license Serial Number** that is visible in **Your licenses** section.
3. In the opened menu select the **Return license** option in order to temporarily disable **CADprofi** license on a given work station. Returning the license will allow other users to use the application.

> ***Hint***
> ***This option allows to temporarily disable the license, restarting the CAD program will allow to download the permissions and restore the application, so it will work again.***
> ***In order to disable the CADprofi program with the network license for a longer period of time, it is recommended to delete the serial number or to separate CADprofi from the CAD program by clicking the Restore CAD program, which is available in the CADprofi configuration window. Configuring the CADprofi application to the CAD program will not require from user to repeat the activation or add the serial number again.***

On-line update

Updating CADprofi program

In **CADprofi**, manufacturer's libraries are being constantly updated, content is being expanded and new features are being introduced. In order to provide users with all the latest updates we have introduced an online update system.
The update process can be done using the **Update now** button, which can be found in the **CADprofi configuration**, **CADprofi - Options** and **CADprofi -About** dialog windows.
It is also possible to enable automatic reminders that will inform about possibility of software update. In order to do this, user has to enable the **Propose checking of updating on-line** that is located in the **CADprofi – Options** dialog window.

Procedures

Updating CADprofi

1. Run the **CADprofi – About** command from the toolbar or from the **CADprofi** menu.
2. (Optional) Run the **CADprofi - Options** command from the **CADprofi** menu or from the **CADprofi** toolbar.
3. (Optional) Run the **CADprofi** configuration program that is located on the Windows desktop.
4. Click the **Update now** button in order to run the **On-line update** system.
5. In the newly opened window click the **Check updates** button in order to connect with the **CADprofi** server and check if there are new updates available. After a while, all newly found updates will be listed in the **Available updates** window.
6. (Optional) In the list of **Available updates** uncheck those items that you don't want to download.
7. Click the **Download** button in order to start downloading selected updates.
8. (Optional) If you have connected to the **CADprofi** server and the update list is empty, then it means that there are no new updates available. Click **Close** in order to close the **Update** window.
9. Click the **Install** button in order to start installing selected **CADprofi** updates.

Notice

CADprofi online update program requires an internet access with an http protocol.
In case of corporate networks, administrators often impose restrictions in internet access. In such cases, CADprofi online update program won't be able to connect to the CADprofi server.

Online updates are available only for users with an active subscription. Subscription can be renewed in our eShop or from CADprofi distributors.

Updating multiple seats

In order to properly carry out the update process it is necessary to download all items at least on one seat. These files are saved in the *Download* folder, which is located in the program main folder (e.g. *C:\Cadprofi*). Because the updating program checks if the necessary files are in *Download* folder before downloading files, user can copy the entire *Download* folder's content into another computer thereby avoiding the necessity of downloading the same update on other seats.

Notice

Regardless of how user obtained the update files, it is still needed to connect to the CADprofi server to determine the program version and subscription validity.

Restore previous version

During the updates installation it is possible to enable the **Create copy of the updated components** option, thanks to which all files that will be copied to the *Archive* folder that is located in the main program folder. Using the **Restore previous version** button will replace the files and return to the previous **CADprofi** version. This option should be used if an error occurred during the update process or if the **CADprofi** application doesn't work properly with the CAD program after the update.

CADprofi® General commands

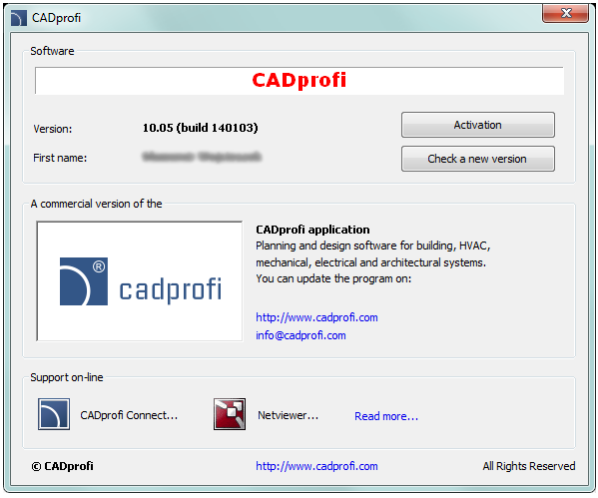
CADprofi General commands

CADprofi program has got many general commands used for numbering and describing objects, creating BOMs, adding your own symbols to the program etc. Editing commands that makes it easier to work on the project are also available.

CADprofi – About command

Basic information about the program version and licensing

CADprofi - About dialog window provides basic information about the currently used program version. The command also allows user to activate his license and register the program, as well as, to run the program update.



CADprofi – About dialog window

Support on–line

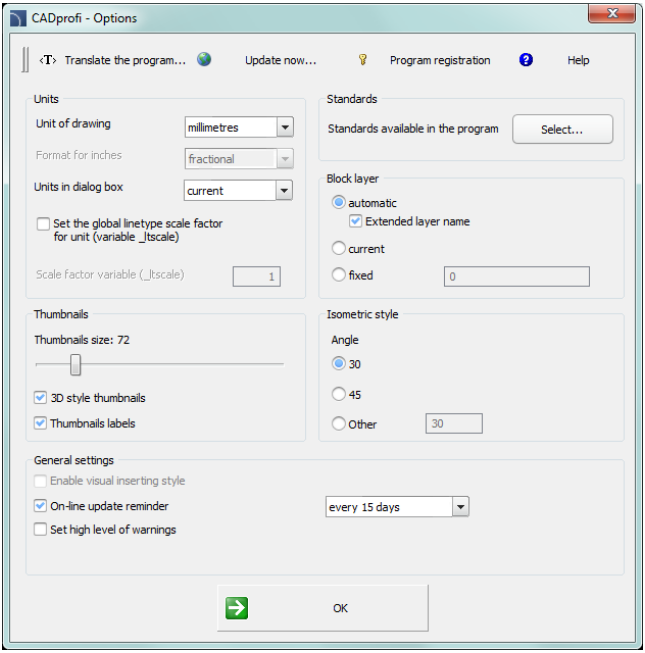
In the **CADprofi - About** window there are buttons thanks to which user can run software that is used for remote diagnostics and on-line help.

After the connection is being established a window is being displayed in which user can see the image from consultant's computer. Thanks to this consultant can explain certain issues related with the use of **CADprofi** software. Voice communication may be conducted by phone or by VoIP call (microphone and headphones are required).

In order to use this tool, user has to call our support department (phone numbers are available on **CADprofi** website) and then turn on the **CADprofi Connect** program. Consultant will give you the ID number and password for the remote session.

CADprofi Options

With the **CADprofi – Options** command it is possible to define the program's basic parameters. In order to open the window, select the **CADprofi – Options** command located on the menu or on the main **CADprofi** toolbar.



CADprofi – Options dialog window

Units

Unit of drawing – determines the current drawing unit for objects inserted from **CADprofi** libraries. Application, depending on the type of chosen units, will insert objects in such sizes to keep the 1:1 drawing rule.

Format for inches – determines how inches will be displayed in **CADprofi**. Both **decimal** and **fractional** formats are available.

Units in dialog box – chooses the type of drawing unit that will be displayed in dialog boxes.

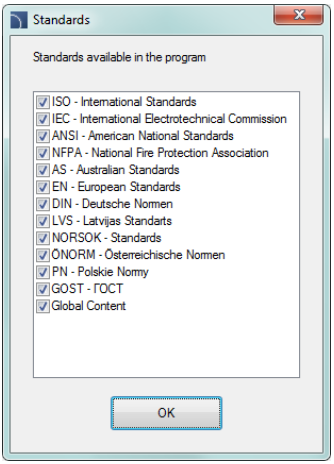
Set the global linetype scale factor for units – this option allows user to automatically adjust the density of lines by adjusting the _LTSCALE variable.

Scale factor variable' (_ltscale) – this option allows user to manually adjust the density of lines by adjusting the _LTSCALE variable.

> ***Hint***
>
> ***If you are designing on a construction project you should check in which unit was it created. For this purpose, you can use the distance command (_DIST) to measure distance between two points in the drawing such as window or door width.***

Standards

Standards available in the program – this option allows user to determine which standards he would like to use. Click the **Select...** button in order to check/uncheck standards available in the program. Deselecting standards from the list means that those standards won't appear in any **CADprofi** library.



Block layer

In **CADprofi** program it is possible to use automatic management of layers. This feature is very convenient because it frees the user from the obligation of placing objects on different layers. **CADprofi** layer structure is used to automatically create BOMs, numbering and in commands that allows to quickly enable or disable layer groups (e.g. quick hide of all objects belonging to the central heating). These commands recognize objects because of their layers. For this reason it is recommended to use automatic layer management system with an extended layer name option enabled, which additionally places objects of different installations on separate layers.

Available layer options:

Automatic – automatic insertion of objects on appropriate layers. Selecting this option allows you to enable the **extended layer name** option (recommended).

Current – allows user to insert objects on a current layer – controlled by user.

Fixed – allows user to insert all objects on a single layer (not recommended). If you have selected this option, you must specify on which layer all objects from **CADprofi** libraries will be inserted.

> ***Notice***
>
> ***All pipe, ventilation and electrical lines are identified only by using layers. Because of this, schematic lines and 2D lines and fittings are always inserted on automatic layers regardless of the selected option.***

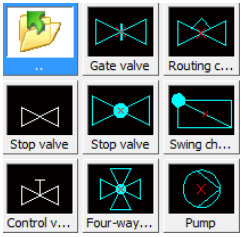
Thumbnails

Thumbnail view is being used in many dialog boxes, because it makes it easier to select items. User can customize the display of thumbnail by changing the following options:

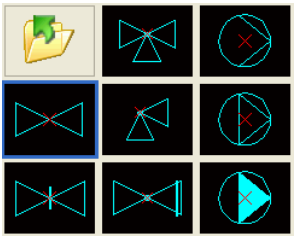
Thumbnails scale – determines the thumbnail size.

3D style thumbnails – this option changes thumbnails style so they look like buttons.

Thumbnails labels – displays the element name on the thumbnail.



Thumbnails with labels, 3D style, size 70

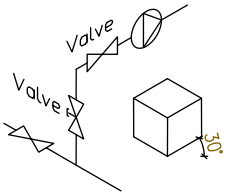


Thumbnails without labels, flat style, size 80

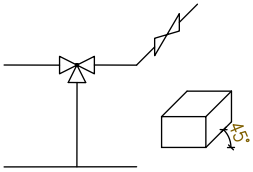
Isometric style

CADprofi can create schematics in both isometric and dimetric view at any angle. Enabling isometric view during object insertion will automatically incline the symbol in accordance with the selected angle.

Angle – determines the isometric symbol's incline angle. Available options: 30° (default), 45°, Other (user can specify his own incline angle).



30° isometry



45° diagonal dimetry °

Additional options

Set high level of warnings – enables /disables warning and critical messages in form of dialog boxes that require user intervention (approval).

Restore typical dialog boxes – this option is visible only when your CAD program has got the FILEDIA or CMDDIA variable switched off. It allows to switch on both those variables and restore the usage of CAD program dialog windows.

Enable 3D blocks – option that allows to switch on the usage of 3D objects. This option is visible only if **CADprofi** has been run on an application that is based on IntelliCAD technology. **Enable 3D blocks** option should be enabled only if your CAD program is capable of using 3D blocks.

Enable visual inserting style – enables/disables the usage of visualization that uses the InputPointMonitor method during insertion of objects such as 2D lines, fittings or channel devices.

Update options

On-line update reminder – enables/disables automatic reminders about the possibility of updating **CADprofi** software. Thanks to this, application won't establish an internet connection automatically, but will only inform the user about such possibility. Automatic reminders can be set to inform about updates every 7, 15 or 30 days.

Update now – this button opens the **On-line update** dialog window in order to check if new updates are available.

CADprofi layers

In **CADprofi** program it is possible to enable the mechanism of automatic layer management. This feature is very convenient, since it relieves user from the necessity of placing objects on appropriate layers. When the automatic layer management option is enabled it is easier to mantain the adequate organization of created drawings, and as a result it is possible to obtain detailed BOMs sorted by industry or type of objects. Appropriate layer structure will also make the printing creation easier.

In order to maintain appropriate line colours and thickness user should use the *CADprofi Color.ctb* or *CADprofi Mono.ctb* plot/printing styles. If CADprofi plot styles are not avilable in the CAD program then user should copy all files from the *C:\Cadprofi\xx\PlotStyle* to the appropriate CAD folder (location of this folder can be checked in the CAD program options e.g. for BricsCAD V13 En installed on the Windows 7 it is the following folder: *C:\Users\Xxx\AppData\Roaming\Bricsys\Bricscad\V13\en_US\PlotStyles*

CADprofi layer structure

Names used for layers that are automatically created by **CADprofi** consist of several parts:

- First 3 characters define the design branch and the type of the created drawing (2D, 3D etc.).
- Next characters define the type of object e.g. a symbol, line, duct etc.
- The ending of the layer name is used to define the belonging of object to a specific installation, type of material, line type, wall etc. (e.g. S-Supply, R-Return, I-Supply air). The ending is created by using the **extended layer structure** (check page 29).

It is also possible to create additional layers in **CADprofi** program such as these that show the drawing details (details, axes, hidings) as well as special layers.

Layer colours

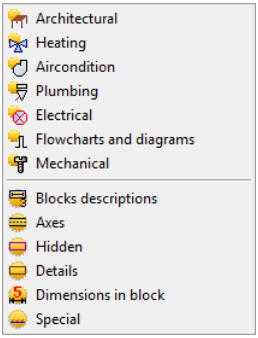
Colours of automatic layer structure are adapted to the **black screen background**. For ergonomic reasons only certain colours from the 10 to 249 range are used. If the colour is defined by a standard then user should use the colour accordingly to the standard, e.g. a supply line used in central heating should have a red colour (or a red tint). For objects whose colour does not depend on the standards user should use colours that makes it convenient to assign them to the print thickness. Plot styles: „**CADprofi Color**" and „**CADprofi Mono**" have got defined 4 line widths. Below is a table of colours for each line width.

Printing colour	**0.15 mm**	**0.35 mm**	**0.5 mm**	**0.7 mm**	**object width**
black	13, 23...	**11, 21...**	15, 25...	17, 27...	19, 29...
colour	16, 26...	14, 24...	**10, 20...**	12, 22...	18, 28...

18, 28 ... 19, 29 ... colours don't have a defined print width (they have got the object width set) and are not used in **CADprofi** program. User can use these colours without fear of conflict with the program's layer structure (colours).

Thaw layer groups

In **CADprofi** program there are commands used to easily enable or disable group of layers, such as all architectural layers or detail layers.



Commands from the **Thaw layer groups** are used to quickly enable a group of layers from a given module or layers that define the drawing detail level (axes, hidings etc.).

Thaw layer group toolbar

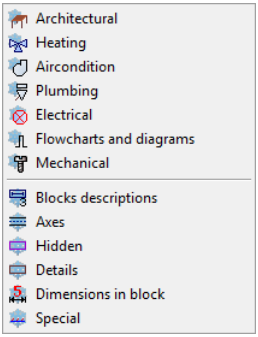
Show all layers – turns on the visibility of all **CADprofi** layers in the drawing.

Show 2D layers only – turns on the visibility of layers that contain 2D objects, layers that contain 3D objects will be frozen.

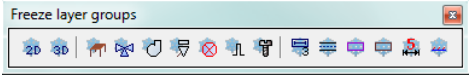
Show 3D layers only – turns on the visibility of layers that contain 3D objects, layers that contain 2D objects will be frozen.

Freeze layer groups

In **CADprofi** program there are commands used to easily enable or disable group of layers, such as all mechanical layers or hidings.



Commands from the **Freeze layer groups** are used to quickly disable a group of layers from a given module or layers that define the drawing detail level (axes, hidings etc.).



Freeze layer group toolbar

Freeze 2D layers – freezes all layers that contain 2D objects.

Freeze 3D layers – freezes all layers that contain 3D objects.

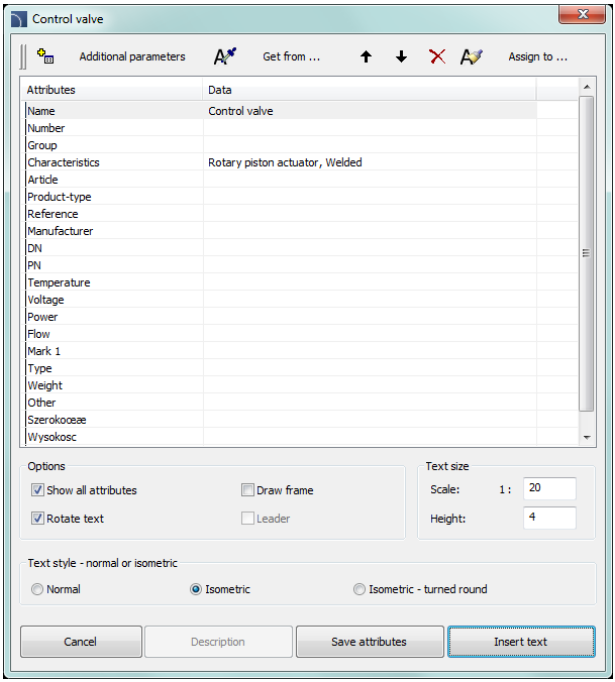
Attributes and descriptions

During the symbols and objects insertion, **CADprofi** program is adding various descriptions and technical parameters to them. This data is stored as attributes. The **Attributes and descriptions** command allows to edit the attributes, as well as to add descriptions (as a simple text) into the drawing. All data that is used in the attributes can be later used during the BOM creation.

After user chooses the command he should indicate an object, whose parameters he would like to edit. This command gives the possibility to edit many objects thank to the **Multiple** function ("**M**" key). As an effect it will be possible to edit parameters of all indicated blocks. After indicating the objects a new dialog window that contains all attributes will appear.

If user has selected several object it's possible that a ***Varies*** value may appear in some data. It means that objects have got different values of a specific parameter. Editing this parameter will also edit it in all of the selected blocks. If this parameter won't be edited then the value won't change in any of the selected blocks.

Attributes and descriptions



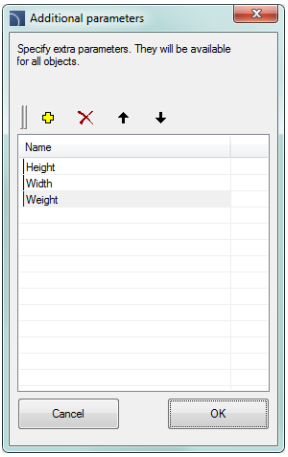
Attributes and descriptions dialog window

The **Attributes and descriptions** dialog window contains the following items:

List of parameters – list that contains all attributes and their values from selected blocks.

Additional parameters– this button opens a new window of additional parameters, that are defined by the user.

- **Add** – adds a new parameter to the list. This parameter will be visible in all program objects as well as in the Bill of materials command.
- **Delete** – deletes the selected parameter. After deleteing the parameter the attributes name will also be erased, although any data will remain as unassigned.
- **Move up** – moves the selected attribute one posiotion up.
- **Move down** – moves the selected attribute one posiotion down.



Recover parameters – allows to recover additional parameters from the unassigned values that are contained in the object. These parameters will receive the default name, that's why it is recommended to change it.

> ***Notice***
>
> ***It is possible to only edit attributes of objects that were inserted to the drawing with CADprofi commands.***

Get from... – allows user to get the selected attributes from the object indicated in the drawing. If the specified object does not have any parameters or their value is empty the operation will not be performed.

Assign to... – allows to assign selected attributes to objects indicated in the drawing. This is a convenient option that's used to agree objects attributes.

Move selected rows up/down – moves the selected attribute up/down one position on the list. This option is needed when user wants to select attributes and insert them in a specific order as a text into the drawing.

Delete contents of selected attributes – clears the data from selected attributes.

Show all attributes – switches between two views: show all available attributes and show only attributes that are filled with data.

Rotate text – enable/disable the ability to rotate the text when inserting it into the drawing.

Draw frame – enable/disable the frame for the text, that will be inserted into the drawing.

Leader – allows to draw a line between the object and the inserted text.

Text size – options that specify the size of inserted text.

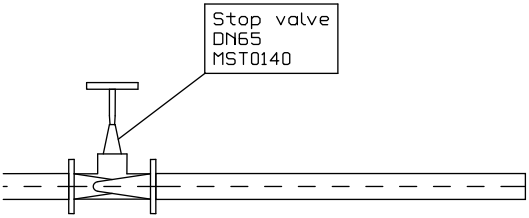
- **Height** – specifies the block text height in millimeters.
- **Scale** – allows to specify the scale in which the drawing will be printed.

Text style: normal or isometric – allows to specify the text style in both normal and isometrical views.

Description – opens window with an optional object description. This function is used only in certain objects from the manufacturers databases.

Save attributes – closes the dialog window and saves all the changes in selected blocks.

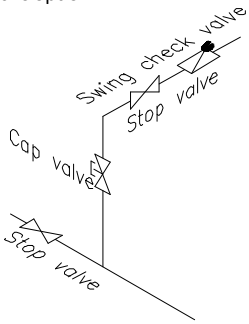
Insert text – saves all the changes made in the selected blocks, and allows to insert to the drawing the value of selected attributes as a plain text. In order to insert the text, user should indicate a point that specifies the first line of text. Depending on the rotation option, it may be necessary to indicate a second point that specifies the text rotation angle. Instead of indicating a second point it's possible to indicate the angle size (e.g. "0" or "90"). In this case, the text will be inserted horizontally. The style of the inserted text depends on the current CAD program style.



Description in axonometric drawings

The **Insert text** command allows to insert texts in axometric drawings. In order to insert text in isometry or dimetry user should select the **Text style** – **isometric** option.

When inserting text in isometry **CADprofi** will activate the isometric mode for the graphic cursor, thanks to which it will be easier to determine the position and text insertion angle along the desired axis. Isometric cursor is automatically set only when the isometric angle is set to 30°. When inserting text in dimetry it is convenient to enter the text angle in the command line. The angle should be set in accordance with the axonometrical angle that is set in CADprofi Options (page 28) dialog window.



Frames and tables

An important part of each drawing is a frame and table.

The **Frames and tables** command offers users a convenient way to insert standard frames and tables to the drawing:

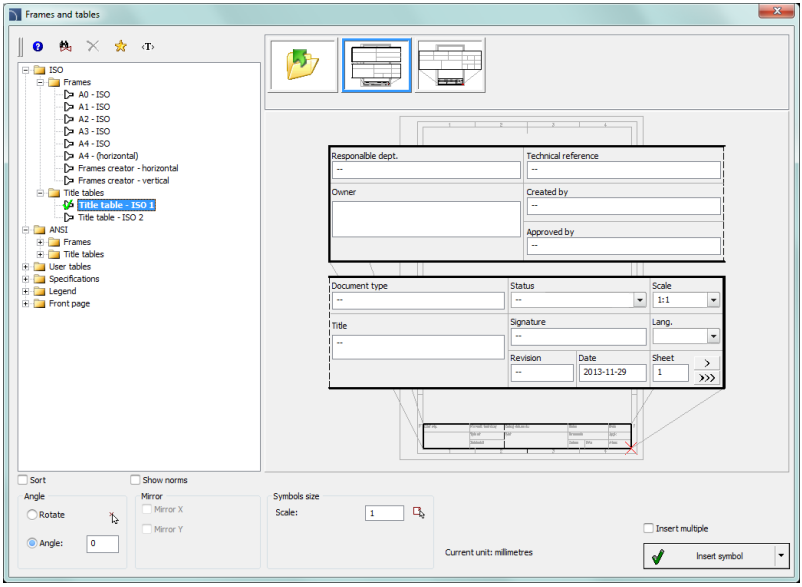
Tables – standard drawing tables.

Specifications – tables used to create drawing specifications.

Legends – standard legends that contain recommended line layers for installations.

User tables – empty tables, which can be defined by the user.

> ***Hint***
>
> ***Legends and drawing specifications can be created using the "Bill of materials ".***



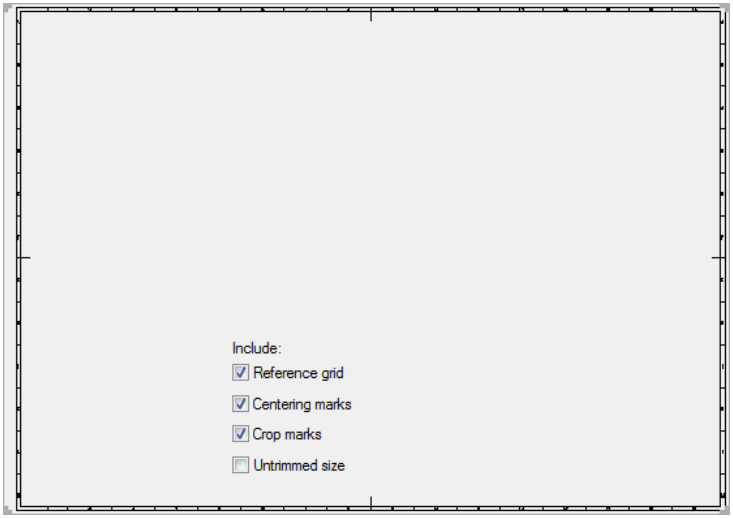
Frames and tables dialog window

The dialog window contains the following items:

Angle/Rotate – gives the possibility to specify a fixed table insertion angle or to enable the rotation option, at which the angle is determined by the rotation.

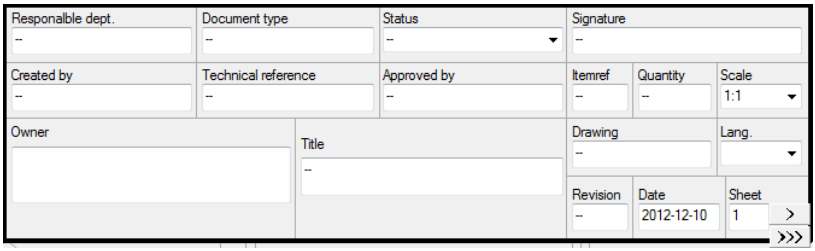
Symbols size – a value that specifies the frame or table scale.

Frames



Frame view can be changed by turning on or off the following options: **Reference grid**, **Centering marks**, **Crop marks** and **Untrimmed size**.

Tables



All tables that are available in the program have got a possibility to edit the information about the project. Some tables are interactive, and for them it is possible to specify all data in the dialog window.

For interactive tables, edition or selection fields (with a pull-down list of typical values under the ▼ button) are available. If in one project (drawing) user inserts many sheets, then the program can enumerate them with the › and ››› button. The **Data** field is filled up automatically.

In order to edit tables after inserting them into the drawing, it's possible to use the **CADprofi Attributes and descriptions** command or an appropriate CAD program command e.g. DDATTE (_DDATTE).

Procedures

Inserting a frame and a table

1. Run the **Frames and tables** command.
2. Select the frame, e.g. **A2 – ISO**.
3. Select the elements that should be included in the frame (**Reference grid**, **Centering marks etc.**).
4. Click the **Insert** button to insert the frame into the drawing.
5. Once again run the **Frames and tables** command.
6. Select a table, e.g. Table – ISO 1.
7. Fill up the data about the project.
8. Click the **OK** button in order to insert the table into the drawing. As the insertion point user should indicate the right, down corner of the previously inserted table.
9. (Optional) Select the Edit symbols command to change the text in the tables or to change table into a different one.
10. (Optional) Change attributes in the table with the help of the CAD program. These changes can be done with the _ATTEDIT command or in the object properties window.

User tables

User can create and add to the program his own tables or frames that are used in the drawing.

Tables that are included in CADprofi program allow user to edit data (blocks attributes). In order for the user tables to have the same functionality, it is necessary to include the appropriate attributes definitions for the created tables.

A convenient way to do this, is to insert and explode a standard table, edit it and afterwards add it as a user block.

The way how to define user own elements is described in detail in the **Defining user blocks** chapter (check page 89).

Numbering

The numbering system allows to assign numbers to symbols and objects that were created with **CADprofi** program. Objects data as well as their numbers is used to create descriptions and specifications. The **Numbering** command can work in two modes. If user runs the command and selects an object or symbol, that hasn't been previously numbered, then the program will allow to number objects. On the other hand if user indicates an object that already has a number, then the command will work in a delete numbers mode.
Program uses separate numbers strings used for various installations, industries and objects types. This means that when user numbers objects, for example from the electrical module, then the program will start the numbering from the number 1. If user starts the numbering process with a different module, then the program will start again from the number 1. To distinguish different numbering strings it is recommended to add prefixes or suffixes to them, so they will look different (e.g. E-1, E-2 for the exhaust system and S-1, S-2 for the air supply).

CADprofi uses the following strings:

- Numbering of 2D lines and fittings for both piping and ventilation installations - separate numbering for the following installations: supply, return, sewage, exhaust, supply air.
- Numbering of 2D electrical trays - separate numbering for different systems.
- Numbering of symbols, objects - common numbering is used for all elements from a particular module.

Notice

The Numbering command is used to add order numbers to individual elements of the project.

CADprofi program also have another methods of numbering:
1. Numbering of electrical apparatus labels.
2. Numbering used for process automation.
3. Numbering (addressing) for electrical circuits.
4. Numbering used for sheets (tables).
5. Numbering used with symbol's marks, which in some cases can be used alternatively to the "Numbering" command.

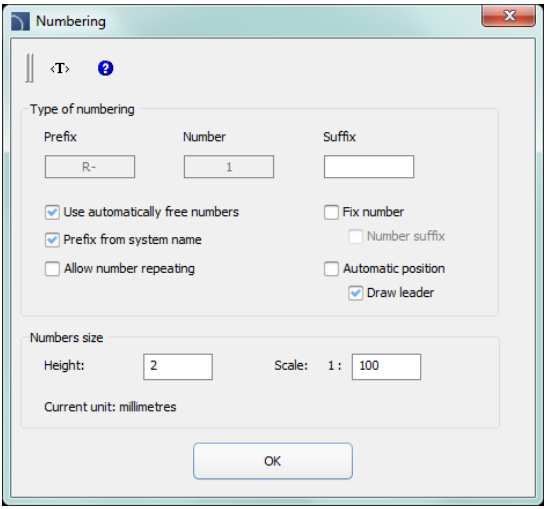
Assigning numbers

In the dialog window user should specify the numbering mode as well as size and appearance of numbers that will be inserted. The number can consist of three parts:

- Correct number must have an integer value from 0 to 32767.
- **Prefix** - any kind of text. Prefix is often used as an indication of an installation or system, e.g. number "A4-12.1" has got a "A4" prefix which means it's a fourth air supply system. The **Prefix from system** name option is used for automatic addition of prefixes.
- **Suffix** – any kind of text. In case of using a multi-level numbering it is possible to automatically add suffixes to them. In order to enable this method user should turn on both **Fix number** and **Number suffix** options. After that **CADprofi** will start numbering only the last of the number.

  Examples of a multi-level numbering (only the last part is being numbered):

  A4–7–**1** A4–7–**2** A4–7–**3**
  A4–7–1.2.**1** A4–7–1.2.**2** A4–7–1.2.**3**
  7–1.2.**1** 7–1.2.**2** 7–1.2.**3**



Numbering dialog window

The dialog window contains the following items:

Prefix – box that allows to add a numbering prefix.

Number – box that allows to specify the object number.

Suffix – box that allows to add a numbering suffix.

Use automatically free numbers – automatically assign numbers to indicated objects. Disabling this option will allow user to manually set the value.

Prefix from system name – automatic completion of prefixes.

Allow number repeating – gives the possibility to repeat object numbers.

Fix number – enables usage of a fixed number for all numbered objects. Enabling this option allows to use suffix numbering.

Number suffix – numbering by adding numbers to the suffix.

Automatic position – automatic placement of a numbering block in a point indicated during the object selection. Enabling this option gives the possibility to specify the placement of the numbering block for each indicated object.

Draw leader – draws a leader between the indicated object and the numbering block.

Numbers size – option that gives the possibility to specify the numbers size on the printing:

- **Height** – specifies the block text height in millimeters.
- **Scale** – allows to specify the scale in which the drawing will be printed.

Numbering

After user runs the **Numbering** command he should indicate an object. If this object doesn't have a number, then a dialog window will appear, in which it will be possible to specify the numbering options. In order to start the numbering process press the **OK** button and depending on settings indicate the numbering placement for the previously indicated object. The command will remain in the numbering mode, allowing user to number additional objects. Within a single numbering command, only objects from the same branch/industry can be numbered. It means that if user starts numbering for example an air supply installation then till he ends the command he will able only to number lines, fittings and objects from the air supply installations. If user wants to number other installations he has to run the command again. In the case of indicating an object, that does not belong to the same branch/industry or if a selected object already has a number then this object won't be numbered e.g.> *Object is not compatible with previously selected object* or > *This object has already got a number: 1.*

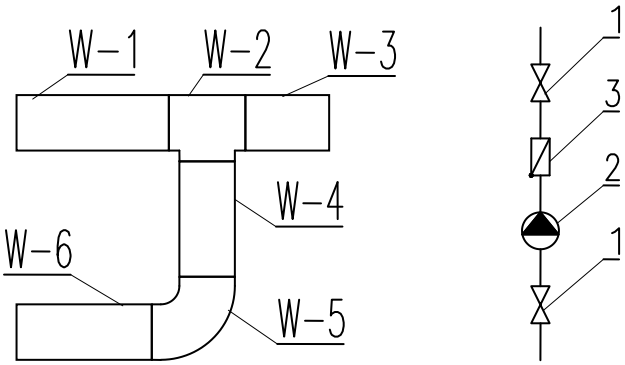
Procedures

Numbering

1. Run the **Numbering** ① command.
2. Indicate the first object that you would like to number, this object will serve as a source object for a given numbering string. The **Numbering** dialog window will appear.
3. (Optional) Specify the numbering options.
4. (Optional) Specify the numbers size on the printing by setting both **Height** and **Scale** for numbering block texts.
5. Click the **OK** button to close the window and start the numbering process.

6. (Optional) If the **Automatic position** option has been turned off then specify the position for the first object number.
7. (Optional) Indicate additional object that you would like to number. If it's necessary you can also indicate numbering blocks.
8. You can end the command by clicking the **Esc** or **Enter** keys.

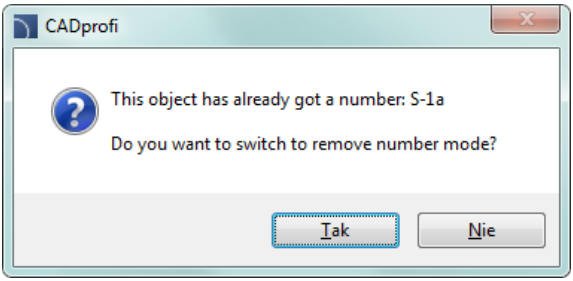
Numbering



Example of different numbering strings in a single drawing

Deleting numbers

During the numbering process to the objects, the numbers are assigned both as a visible text and as an invisible data. Deleting text with the number won't delete number from the object. For this reason, in order to delete numbers, user should use the **Numbering** command.
If user selects after he runs the command, a block that contain information about numbering then a window will appear, that allows to enable the remove numbers mode.

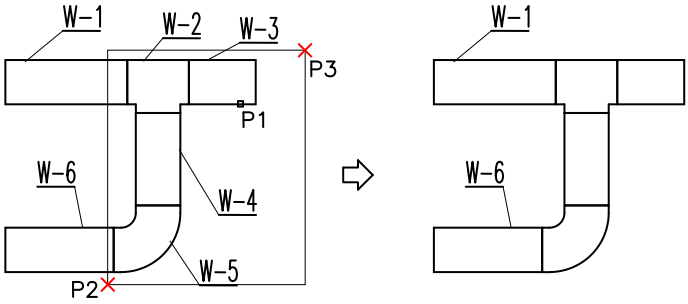


After user activates the remove numbers mode, he should to select desired objects or numbers. Internal information as well as number texts and leaders will be removed.

Procedures

Deleting numbers

1. Run the **Numbering** command.
2. Select an object that contains numbers (**P1**). A dialog window will open.
3. Choose **Yes**, to start the process of removing numbers.
4. The number of the first selected object will be deleted.
5. (Optional) Indicate additional objects (**P2–P3**), from which you would like to remove numbers.
6. End the remove numbers command by clicking the **Esc** or **Enter** key.



Notice

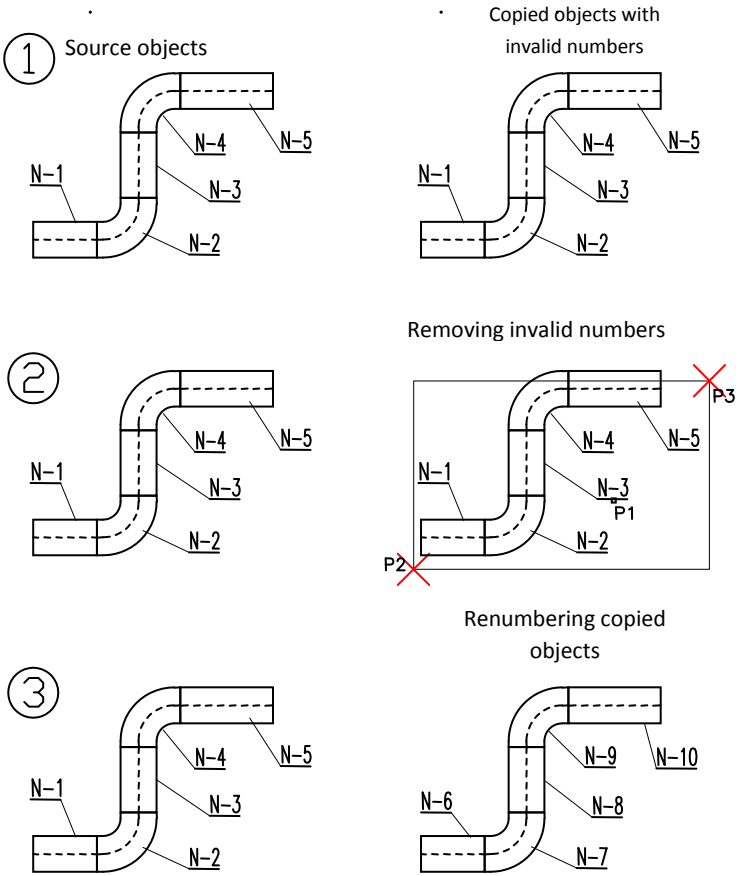
Numbering is being removed from objects when user:
1. Inserts equipment in 2D lines or ducts.
2. Connects some certain lines or ducts.
3. Runs the "2D Divide pipes/ducts command".
4. Removes the symbols by using the "Delete symbols" command.

Copying objects with numbers

After copying, user should remove invalid numbers and renumber the copied objects again. In order to delete these numbers user can use the remove numbers mode from the **Numbering** command. After user runs the command he should indicate the invalid text (**P1**), and then with the help of the selection window delete texts (**P3–P4**). Other objects may be present in the selection window, as the command will only remove numbers.

Notice

Very important: ***When user copies numbered objects the source elements are treated as objects without numbers!!!***

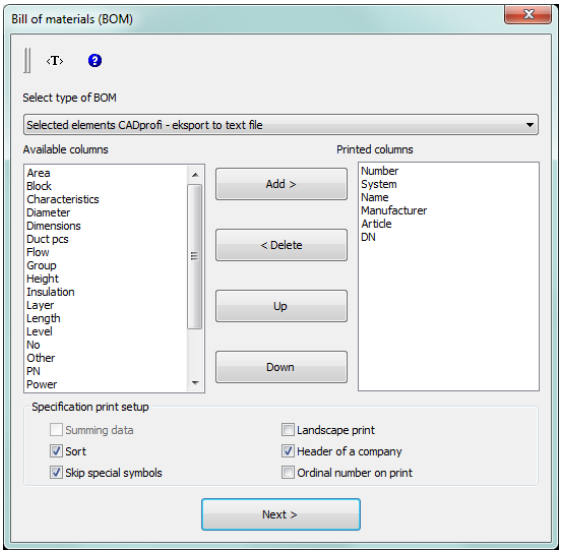


Removing invalid numbers and renumbering of copied objects

Bill of materials

The **Bill of materials** command allows to obtain various data extracts and printings. The easiest and fastest way to receive a specification is to select one of templates and to directly print it. There are several pre-designed templates to choose from, and they are used to obtain various specifications. Depending on the selected template, **CADprofi** performs filtering of selected objects. For instance, when user selects specifications for symbols, **CADprofi** will select only symbols.

Each template provides different data that can be printed in tabular layouts. All needed data are available on the left side of the dialog window. Columns that will be printed are located on the right side of the dialog window. It is possible to select data to the list of printed columns as well as to change the column sequence. BOM can be printed in both horizontal (landscape) and vertical (portrait) mode. For horizontal (landscape) mode, user can select more columns.



Bill of materials (BOM) dialog window

The dialog window contains the following items:

Select type of BOM– a list that contains pre-design templates. User can adapt the **Objects, symbols and lines CADprofi** template to his own needs. Definition of this template is located in the *C:\Cadprofi\X.x\Common\Extract\Extrac.utc* file. In this file it is possible to names of attributes and additional data that is used in BOMs. Futher information can be found in comments in the mentioned file.

Available columns – list of attributes, which values can be used in the printed table. The list is dependant on the chosen type of BOM.

> ***Hint***
>
> ***On the left side of the dialog window there are additional parameters that can be defined by user.***
> ***Thanks to this, it is possible to use these parameters in BOM - „Additional parameters" (check page 35).***

Printed columns – list of attributes that will be included in the columns of the printed table. This list can be later edited by the user.

Add all – adds to list of printed columns all available BOM columns.

Add – adds the selected attribute to the list of printed columns.

Delete – removes the selected attribute from the list of printed columns.

Delete all – removes the all positions from the list of printed columns.

Up, Down – allows user to specify the positions order on the list of printed columns.

Summing data – enable/disable the creation of a collective bill of materials with summing data of used identical objects. It is also possible to include the line length, ducts area, fittings data etc. in the summing data.

Sort – enable/disable data sorting on printings. Data can be sorted in the next dialog window that contains the initial printing preview.

Skip special symbols – enable/disable the inclusion of blocks that haven't got the **NAME** attribute or the ones for which the **NAME** attribute doesn't have any value. Also in BOMs, blocks that are used to describe the drawing (slope symbols, number of core electric wires etc.), are not included.

Landscape print – enable/disable the creation of horizontal (landscape) printings.

Header of a company – enable/disable the printing of the header of a company on the first page of BOM. The data needed for the header and the company logo are being defined in the next dialog window that contains the initial printing preview.

Ordinal number on print – enable/disable the addition of the column with the order number.

Next – confirms the selected options and proceeds to the selection of objects on the viewport. To select them, user can use all kind of methods in the CAD program. The application itself will filter objects according to the chosen template. Thanks to this instead of selecting single objects, it's possible to indicate an area with two points.

Collective BOMs

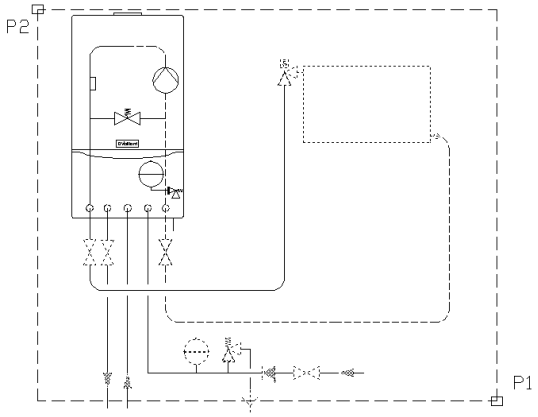
In addition to obtaining bill of materials from the current drawing, it is also possible to create BOMs for projects that contain several drawings. In order to create such a specification, user should make a temporary file in which data from all drawings will be saved. Afterwards, this file can be printed as a collective BOM.

In order to create a file with temporary data, user should press the **Save data** button. Afterwards user should specify the file, in which the data will be saved. If user specifies an existing file then the data will be added to the data that already exists. To create a collective data it is necessary to run the **Bill of materials** command in each drawing, and to select the same template and save the data to the same file. To print the data from the temporary file, user can use the **Open data** button.

The temporary file has got a csv format structure, therefore this file can be used to export data for example to a spreadsheet.

Data window

Data window is opened after user has indicated objects.

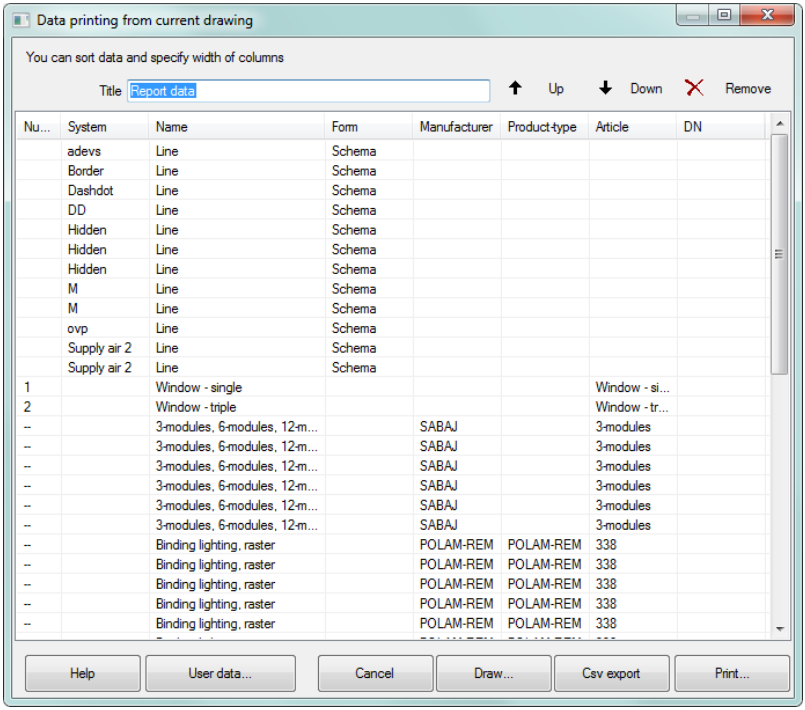


Example of object indication by using two points

In this data window it is possible to sort the data, set the column width that will be used for printing and delete any rows or columns or hide unnecessary data.

To sort the data, user should click on the title of the desired column. Clicking again will reverse the sort order. User can sort multiple columns. In this case, user should first sort columns with the lowest sort priority.

To specify the column width, user should set the cursor between the columns (in the header). Cursor will change into a double-headed arrow that will allow to change the width by dragging. In this way it is possible to completely hide selected columns. This option is useful to hide empty columns or columns that are not needed for printings.
In the preview window user should specify the printing title. It is also possible to define the printing header such as: contact details and company logo.



Nu...	System	Name	Form	Manufacturer	Product-type	Article	DN
	adevs	Line	Schema				
	Border	Line	Schema				
	Dashdot	Line	Schema				
	DD	Line	Schema				
	Hidden	Line	Schema				
	Hidden	Line	Schema				
	Hidden	Line	Schema				
	M	Line	Schema				
	M	Line	Schema				
	ovp	Line	Schema				
	Supply air 2	Line	Schema				
	Supply air 2	Line	Schema				
1		Window - single				Window - si...	
2		Window - triple				Window - tr...	
--		3-modules, 6-modules, 12-m...		SABAJ		3-modules	
--		3-modules, 6-modules, 12-m...		SABAJ		3-modules	
--		3-modules, 6-modules, 12-m...		SABAJ		3-modules	
--		3-modules, 6-modules, 12-m...		SABAJ		3-modules	
--		3-modules, 6-modules, 12-m...		SABAJ		3-modules	
--		3-modules, 6-modules, 12-m...		SABAJ		3-modules	
--		Binding lighting, raster		POLAM-REM	POLAM-REM	338	
--		Binding lighting, raster		POLAM-REM	POLAM-REM	338	
--		Binding lighting, raster		POLAM-REM	POLAM-REM	338	
--		Binding lighting, raster		POLAM-REM	POLAM-REM	338	
--		Binding lighting, raster		POLAM-REM	POLAM-REM	338	

Data printing from current drawing dialog window

Print – allows to print or export the data into many formats.

Csv export – allows to instantly export the data into a *csv* file, which can be loaded into applications such as MS Excel, Open Office or others.

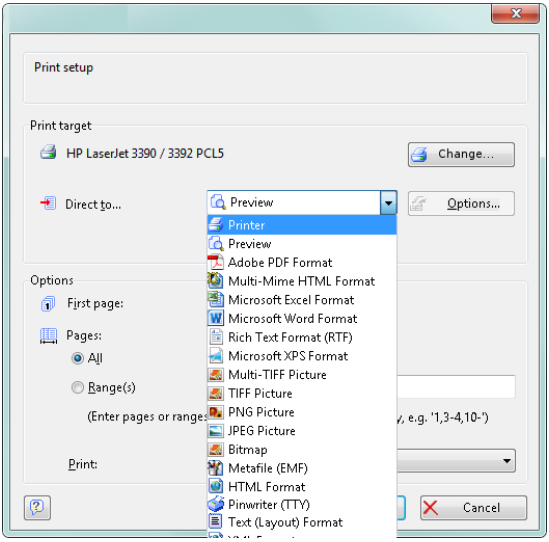
Draw – is used to create graphical legends and for automatic filling of specification tables.

User data – allows to specify the contact details and the company logo.

Remove – allows to delete rows that contain data. It is also possible to delete data by pressing the **Delete** button.

Printing/Data export

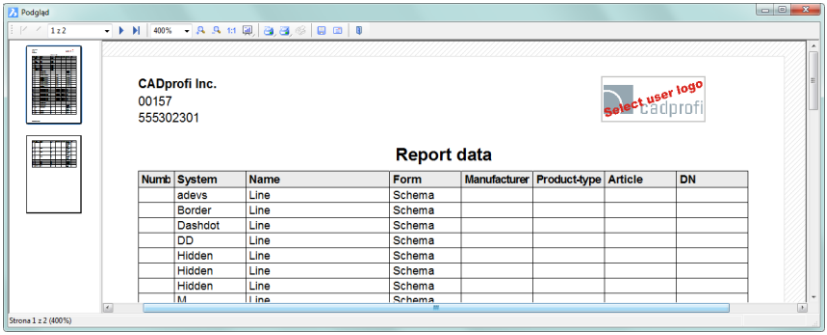
After clicking the **Print** button a window will appear with printing options.



Printing options dialog window

It is possible to directly print the document on a chosen printer or to export the data to various formats (xls, txt, rtf, pdf, html, xml and many more).

A useful option is a possibility to print the first page of specification on another printer than other pages. This option can be used to print the first page that contain the company logo on a color printer and other pages on a monochrome printer.



Printing preview dialog window

When user exports data it is possible to specify the export parameters such as only export the data, or export data with used formatting style.

> ***Hint***
>
> ***Printing preview can be saved in files with an LL extension. To view these files, user can use the "LLview11.exe" program, which is located on the disk, in the CADprofi main folder (e.g. "C:\CadProfi\X.x").***
>
> ***Direct printing is not available in some 64bit CAD programs.***

Procedures

Printing specification

1. Run the **Bill of materials** command.
2. Choose a template from the list of available templates in order to determine the BOM type.
3. (Optional) Add or delete a position in the **Printed columns** panel.
4. (Optional) Choose the printing option for specification.
5. Click the **Next** button.
6. Select objects from the drawing that you would like to include in the specification. Confirm by pressing the right mouse button or the **Enter** key.
7. (Optional) Change settings in the **Data window**.
8. Click the **Print** button.
9. In the newly opened **Printing options** window in the **Export medium** card select the printer type that you would like to use.
10. (Optional) Change settings in the **Printing options**.
11. Click the **Start** button to begin printing.

Data export (Excel)

1. Run the **Bill of materials** command.
2. Choose a template from the list of available templates in order to determine the BOM type.
3. (Optional) Add or delete a position in the **Printed columns** panel.
4. (Optional) Choose the printing option for specification.
5. Click the **Next** button.
6. Select objects from the drawing that you would like to include in the specification. Confirm by pressing the right mouse button or the **Enter** key.
7. (Optional) Change settings in the **Data window**.
8. Click the **Print** button.
9. In the newly opened **Printing options** window in the **Export medium** card expand the list **Directly to…** and select **Microsoft Excel Format**.

10. (Optional) Click the **Options** button and go the **Result** card. It disables the Export data only from objects table option to save formatting.
11. (Optional) Change settings in the **Printing options**.
12. Click the **Start** button to begin the data export.

Creating specifications (Mechanical)

Creating specifications

CADprofi Inc.
00157
555302301,00

Report data

Numbe	Name	Manufacturer	Article	DN	PN
--	Brenner	Viessmann			
--	Brenner	Viessmann			
--	Kessel	Viessmann	Vitoplex oder Vitomax		
--	Kessel	Viessmann	Vitoplex oder Vitomax		
--	Regler Vitotronic	Viessmann	Vitotronic		
--	Regler Vitotronic	Viessmann	Vitotronic		
--	Regler Vitotronic	Viessmann	Vitotronic		
--	Regler Vitotronic	Viessmann	Vitotronic		
--	Warmwasserspeicher Vitocell	Viessmann	Vitocell V		
--	Heizung Heizkreispumpe	Vaillant			

Exporting to Excel with preserved formatting style

> ***Hint***
>
> ***It is also possible to export the data to the Excel program with the „Export csv" button.***

Creating legends and specifications

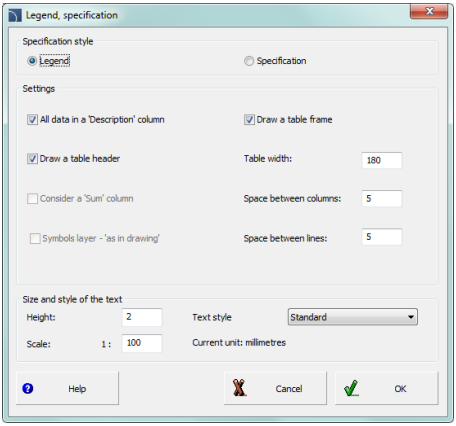
In order to create a specification or a legend with the help of the specification wizard user should press the **Draw** button. Two modes are available: **Legend** and **Specification**.

The **Legend** mode is used to create any defined legends or specification tables. In this mode it is possible to determine the overall table width, the spacing between columns and rows and the text style. The legend may contain a header with a specification title and the titles of each column.
The **Description** column is particularly important. After enabling the option: **All data in the 'Description' column**, the legend will be created in which all data that will be acquired from the drawing will be consolidated into one description column.
When disabling the above option, it is possible to create tables that contain many columns. Legends that contain symbols and objects views are created only if the **Block** column has been selected in the specification wizard. In this case symbols that are in the legend can be

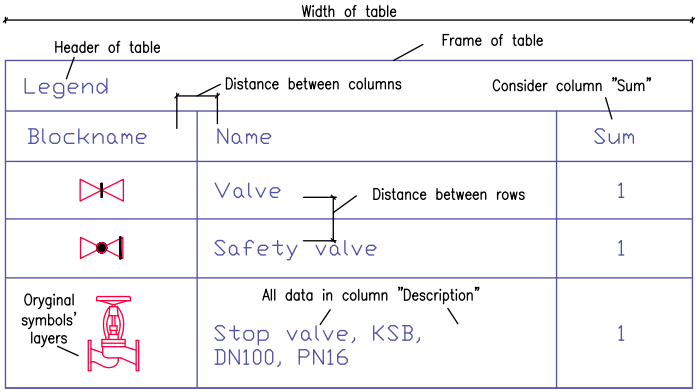
placed on layers accordingly to symbols layers in the drawing. Thanks to this, symbols that are in the legend have got saved colours and characteristics of the original objects from the drawing.

> ***Notice***
>
> ***Legends that contain symbols are created only on the basis of the current drawing (it is not possible to create a collective graphical legend from several drawings).***

Legends columns width is determined in proportion to the column width that is set in the main data window. After setting the mouse cursor between the columns, the cursor changes into a double-headed arrow that allows to change the width.



Legend, specification dialog window



Procedures

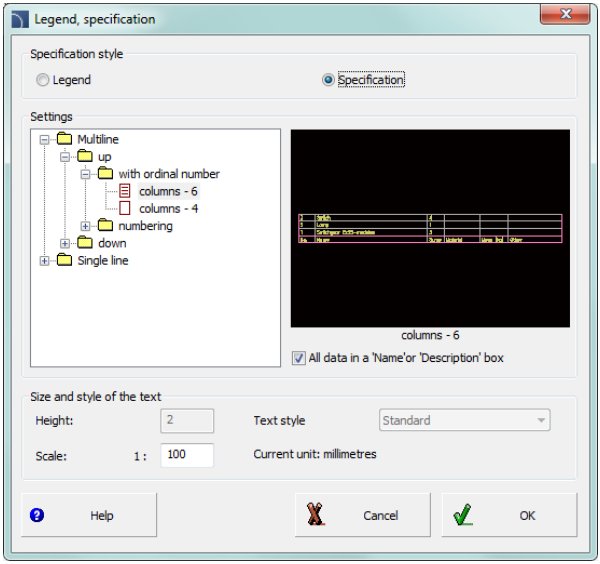
Creating a legend

1. Run the **Bill of materials** command.
2. Choose one of available templates, in order to specify the BOM type.
3. (Optional) Add or delete positions in the **Printed columns** chart.
4. (Optional) Choose the printing option for specification.
5. Click the **Next** button.
6. Select objects from the drawing that you would like to include in the specification. Confirm by pressing the right mouse button or the **Enter** key.
7. (Optional) Specify the settings in the data window.
8. Click the **Draw** button.
9. In the newly opened **Legends, specifications** window choose the **Legend** option.
10. (Optional) Specify the specification table settings.
11. (Optional) Specify the text size and style.
12. Click the **OK** button.
13. (Optional) Select **Yes/No** in the newly opened dialog window in order to delete/save the existing legend.
14. Specify the legend insertion point.

Creating graphical legends

The **Specification** mode is used to fill template specification drawing tables. These specifications are prepared on the basis of appropriate standards. Because of that the specifications tables have got a precisely defined structure (column width, appropriate text size and style etc.). When inserting a specification table into the drawing, user should make sure that he has chosen the right columns with data in the specification wizard, that belong to the appropriate table type. Otherwise, the tables won't be properly filled. The **All data in the 'Name' or 'Description'** box allows to consolidate the data in the column that contains a name.

> ***Notice***
>
> ***There is a set of names of each parameter in the program such as: "Name", "Type", "DN", "PN" etc. In certain tables it is necessary to use other names, e.g. the parameter "Type" in some tables is the equivalent to the "Symbol" or 'Marking" parameter.***



Legend, specification dialog window

Procedures

Creating a specification

1. Run the **Bill of materials** command.
2. Choose one of available templates, in order to specify the BOM type.
3. (Optional) Add or delete positions in the **Printed columns** chart.
4. (Optional) Choose the printing option for specification.
5. Click the **Next** button.
6. Select objects from the drawing that you would like to include in the specification. Confirm by pressing the right mouse button or the **Enter** key.
7. (Optional) Specify the settings in the data window.
8. Click the **Draw** button.
9. In the newly opened **Legends, specifications** window choose the **Specification** option.
10. (Optional) In the **Settings** chart select the **Specification** type.
11. (Optional) Enable/disable the **All data in the 'Name' or 'Description' box** option.
12. (Optional) Change the text size and style.
13. Click the **OK** button.
14. (Optional) Select **Yes/No** in the newly opened dialog window in order to delete/save the existing specification.
15. Specify the specification insertion point.

9		Druckbegrenzer max.	2
8		Dreiwegeventil	5
7		Außentemperaturwächter	1
8		Absperrventil	33
5		Heizung Heizkreispumpe	1
4		Warmwasserspeicher Vitocell	1
3		Regler Vitotronic	4
2		Kessel	2
1		Brenner	2
No	Symbol	Description	Sume

Responsible dept. ---	Technical reference ---	Document type ---	Document status Released	Scale 1:1
	Created by ---	Title ---	Signature ---	Lang. GB
	Approved by ---		Revision --- Date 2013-02-15	Sheet 1

An example table contains columns with **No.**, **Symbol (type)**, **Description**, **and Sum**. In this case in the specification wizard user should choose the Type and Name columns. **Sum** column is being filled only if the **Summing data** option is enabled. **No.** column is filled only if the **Ordinal number on print** option is enabled.

> ***Notice***
>
> ***Blocks that are included in the tables are used only for the purpose of symbol/object illustration. They don't possess the data needed for proper use in CADprofi project. For this reason it is not recommended to copy blocks from the drawing.***

Creating line and cable specifications

In order to successfully create a specification that contains schematic lines user should select the appropriate template that filters objects (Plumbing/heating/air condition/electrical - lines). In this type of specifications, **CADprofi** program includes cables drawn as lines or polylines, as well as sections of vertical lines drawn as risers' symbols.

Procedures

Creating a line specification

1. Run the **Bill of materials** command.
2. Select the template that is used for schematic lines for the appropriate branch.
3. (Optional) Add or delete positions in the **Printed columns** chart.
4. (Optional) Enable the **Summing data** option in order to have the visible **Sum** column in the specification.
5. Click the **Next** button.
6. Select objects from the drawing that you would like to include in the specification. Confirm by pressing the right mouse button or the **Enter** key.
7. Print or insert into the drawing the created specification.

Creating specifications of lines, ducts and 2D cable trays

Fittings and air condition ducts with a rectangular cross-section that are prepared from steel are usually connected with the use of flanges. In **CADprofi** such elements are simplified (without flanges) therefore **user should specify in the project the type or the technology used in the air condition lines. An appropriate annotation should be included in the descriptive section.** If necessary, it is possible to insert flanges into the drawing with the use of appropriate **CADprofi** command. Designer can also decide whether the flanges inserted in the drawing should be included in the specification.
The simplified drawings allow to use most lines and fittings from the **CADprofi** program in any installation projects that use different technologies, materials and connection methods (e.g. metal and artificial/plastic lines installations etc.).

During the specification creation it is possible to add the area of lines and fittings. The area is being calculated on the basis of the PN-EN 14239 standard.
In specifications it is possible to add dimensions or types (markings) of fittings or lines. Fitting type is available only in the case of using elements that are chosen from manufacturers' databases. In another case type is not specified and in the specifications only the main dimensions are included. These dimensions are appropriate with markings that are included in the catalog of 2D fitting drawings. We propose to print and add **the catalog of 2D fittings** to the specification. The catalog of 2D fittings can be found in the *Fittings.pdf* folder that is located in the *file_gb* folder, which can be found in the **CADprofi** installation folder.

> ***Hint***
>
> ***Add the „Fittings.pdf" printing to the fittings specification.***

Cable trays, cable ladders, protective tubes and other elements are available in the Electrical module. Some fittings that are in the **2D fittings catalog** are not used in electrical installations and therefore they are not available in the Electrical module.
Flexible cables with a circular cross-section (FLEX) or flexible protective tubes from the Electrical module are drawn with the help of straight sections and fittings. Still fittings are being treated as lines/cables therefore in specifications all flexible elements are being shown as flexible lines with specified length.

In order to include length and the amount of each line/duct/cable tray sections in the specification user should add both **Length** and **Ducts pcs** (number of pieces) columns.
In order to add only the total length of line/duct/cable tray with the same cross-section in the specification, user shouldn't include the **No.**, **Length** and **Ducts pcs** columns.

An example of a specification of air conditioning installation elements:

This column includes the number of lines/ducts sections with the same length.
E.g. 6 sections with an 1800 mm length.

In the **Length** column the element length is specified in **mm**.
E.g. sections with a 1800 mm length

System	Name	Article	Duct pcs	Length	Area	Sum
Supply air	A pressed saddle	D2=160, D=200, L=200, a=90?°		200	0.151	1.00
Supply air	Duct	D=160	1	918	0.461	0.92
Supply air	Duct	D=200	2	1800	2.262	3.60
Supply air	Duct	D=300	1	1675	1.579	1.68
Supply air	Duct	D=300	1	814	0.767	0.81
Supply air	Duct	D=300	1	904	0.852	0.90
Supply air	Duct	D=300	6	1800	10.176	10.80
Supply air	Duct	D=300	1	91	0.859	0.91
Supply air	Duct	D=300	1	1036	0.976	1.04
Supply air	Elbow	D=300, R=450, a=90?°, E,F=50			1.884	2.00
Supply air	Tee-piece	D=300, D3=300, L=400, L3=200, a=90?°		400	0.565	1.00
Supply air	Tee-piece branch	D=300, D2=200, D3=200, L=423, XY2=250, XY3=300,...		423	0.961	1.00

The **Area** column contains the element area in **m²** or the total area of all elements that were included in the specification.

The **Sum** column contains the total length of lines / trays / ducts in **meters**. The **Sum** column is shown only if user has enabled the **Summing data** option during the specification creation.

System	Name	Article	Area	Sum
Supply air	A pressed saddle	D2=160, D=200, L=200, a=90?°	0.151	1.00
Supply air	Duct	D=160	0.461	0.92
Supply air	Duct	D=200	2.262	3.60
Supply air	Duct	D=300	15.209	16.14
Supply air	Elbow	D=300, R=450, a=90?°, E,F=50	1.884	2.00
Supply air	Tee-piece	D=300, D3=300, L=400, L3=200, a=90?°	0.565	1.00
Supply air	Tee-piece branch	D=300, D2=200, D3=200, L=423, XY2=250, XY3=300, E,F=20	0.961	1.00

Fittings

Fittings types or dimensions are shown in the Article column. All dimensions are the same as those that are used in the drawings from the catalog of 2D fittings.

In the air condition installations it is possible to add the **Area** column that contains area (in m^2) of elements that are counted accordingly to the PN–EN 14239 standard.

For fittings the **Sum** column contains the total amount of fittings from each type.

In the above example there are two 90º elbows in the specification with an ø300 diameter and R=450 (1,5D) radius.

CADprofi – Tools

Editing commands are located in the upper program menu, as well as on a separate **CADprofi – Tools** toolbar. These commands are universal and they work with objects from all **CADprofi** modules. To ensure a fast access to these commands it is recommended to enable the editing toolbar.

CADprofi – Tools toolbar

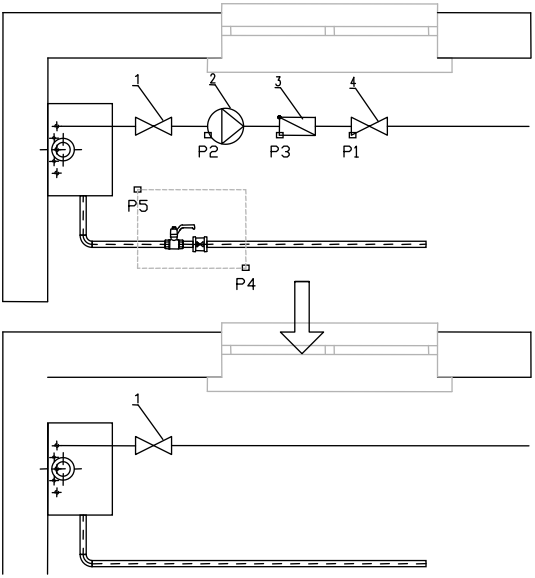
Delete symbols

The **Delete symbols** command is used to delete symbols and armature. When deleting elements that were inserted in lines, program will automatically connect these lines. When deleting numbered symbols, numbers will also be deleted. It is possible to indicate many drawing elements; still the program will delete only these symbols and objects that were inserted with the use of **CADprofi** commands.

Procedures

Deleting symbols

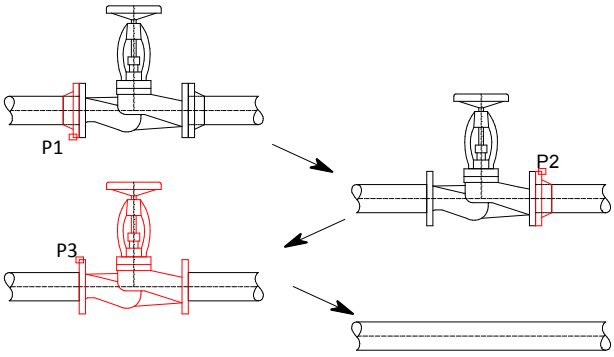
1. Run the **Delete symbols** command.
2. Indicate a symbol that will be deleted from the drawing or line (**P1**).
3. (Optional) Select the **Multiple** option to indicate many objects (**P4**–**P5**). Confirm by clicking the **Enter** button.
4. Indicate additional elements (P2, P3) or finish the command by clicking the **Enter** or **Esc** key.



Procedures

Deleting armature

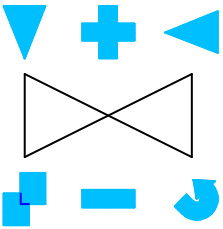
1. Run the **Delete symbols** command.
2. Click the first element (**P1**). It will be deleted, and in its place the line will be lengthened.
3. (Optional) Click additional elements (**P2**, **P3**).
4. End the command with the right mouse button or the **Enter** key.



Quick edition

The **Quick edit** command allows to perform basic editing operations on symbols and objects. After running the command, user should indicate the symbol that he would like to edit. Near the symbol, additional markers will appear that allow to instantly make the necessary changes.

Markers that are displayed after indicating the object:



Mirror – markers that allow to perform symbol mirroring.

Scale – allow to increase or decrease the symbols size on the drawing. The sign increases the symbol's scale by 25%. The sign reduces the symbol's scale by 25%. If the current scale value is 200, then the markers will increase/decrease the symbol's scale by 50.

Copy – allow to quickly create multiple copies of the indicated object.

Rotation – allow to rotate the symbol around the insertion point. If the symbol is located in the line then the application will only allow to rotate it by 180°.

Lengthen – this marker is used to lengthen or shorten normalized elements e.g. screws, rivets, etc. (check page 299).

Quick edit is often used after user adds several symbols into the drawing e.g. with the **Multisymbols** command. If several symbols are being inserted then it is possible that some symbols may be inserted in the opposite way. Quick edit will allow to quickly rotate them in the proper way.

The **Quick edit** command is also useful to turn objects, e.g. when it's needed to change the direction of duct devices or armature.

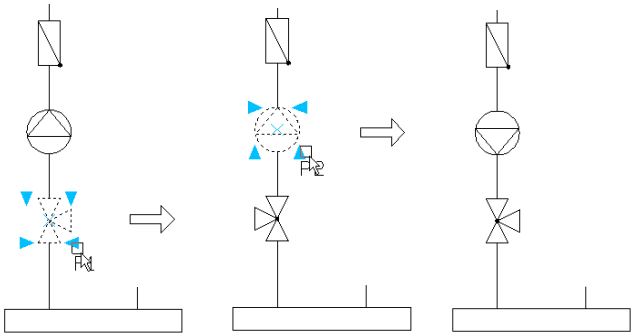
Procedures

Quick edition

1. Run the **Quick edit** command.
2. Indicate a symbol that will be edited.
3. (Optional) Rotate the symbol with the **Mirror** command (**P1**, **P2**).
4. (Optional) Increase/decrease the symbol by clicking the markers.

5. (Optional) Rotate the symbol by clicking the rotate marker.
6. (Optional) Click on an additional element that you would like to edit.
7. (Optional) Insert a copy after selecting the copy marker.
8. End the command with the right mouse button or the **Enter** key.

Quick edition

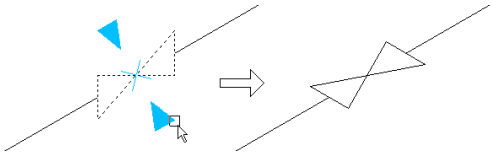


Notice

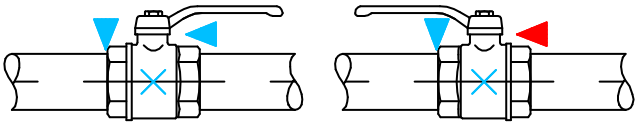
„Mirror", „Scale", and „Rotate" can be used repeatedly, because they don't end the „Quick edit" command. After user copies the symbol the command is being completed.

Available options depend on the type of edited object.

The quick edit command also allows to edit isometric symbols.

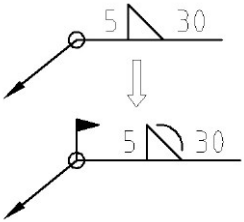


An example of a rotated armature.

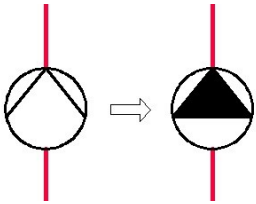


Symbols edition

The **Edit symbols** command allows to freely edit **CADprofi** symbols or objects. This command is particularly important when editing multivariant symbols such as: welding marks, roughness marks, armature, apparatus and many more.
To change the symbols, select the **Edit symbols** command and choose the desired symbol. A dialog window will appear which was used to insert the indicated symbol. Make all the necessary changes in the dialog window. After clicking the Insert, Apply or OK button the symbol will be updated on the drawing accordingly to the changes that we made.
The **Edit symbols** command also allows to convert a symbol in an alternative one.



Symbols edition



Symbols conversion

Symbols edition

Insert similar object

During projects creation it's often needed to insert symbols or objects from many libraries and categories. Because **CADprofi** program has got a very big content, choosing the right element may require excessive searching of libraries or changing categories. In order to speed up work, user may use the **Insert similar object** command.

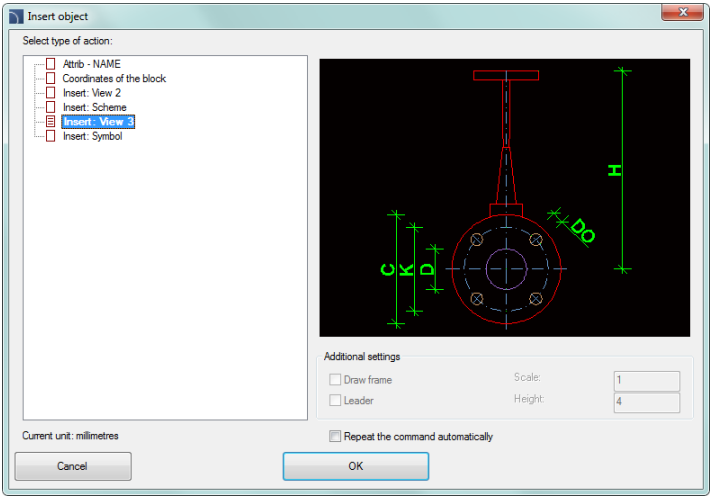
After selecting the **Insert similar object** command user should indicate an element on the drawing. Program will open this element in its original dialog window.

This command makes it easier to insert alternately both one pole and multi pole symbols when creating electrical schemes. In the HVAC & Piping module the Insert similar object command can be often used to alternately armature symbols and devices from different categories into the drawing.

The **Insert similar object** command can be used for all **CADprofi** program elements, including HVAC, air condition, furniture, construction and other elements.

Create alternative view

In **CADprofi** there are many objects that have got the possibility to select the insertion view. Depending on the selected object certain 2D views will be available and for some manufacturers' libraries also 3D views. The **Create alternative view** command allows to quickly place one of available alternative views. When using this command, a new block is being inserted into the drawing.



Create alternative view dialog window

The dialog window contains the following items:

Select type of action – list of available views or procedures for selected block.

Additional settings – settings for some elements that contain texts.

Repeat the command – enable/disable the possibility of re-inserting the element for selected repeatedly inserted objects (omitting the dialog window).

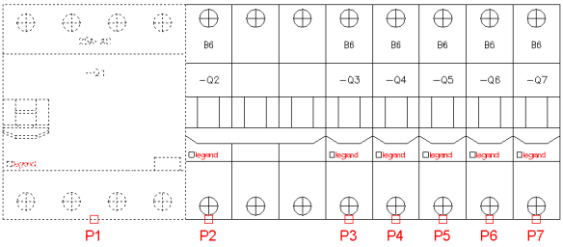
Creating schemes with the Create alternate view command

The **Create alternative view** command allows to insert symbols, that are associated with devices views. It allows to quickly create electrical installations schemes on basis of switchgear views or vice versa.

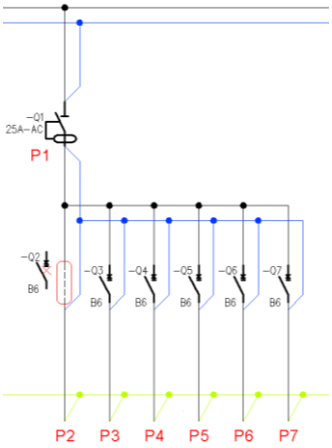
After running the command, user should select the desired apparatus (or symbol). In the next dialog window, he should choose an alternative view, which will be inserted into the drawing. It is recommended to enable the **Repeat the command automatically** option.

Symbols can be inserted into existing lines. In this case, program will automatically break them.

The below pictures show a quick way to create the scheme on the basis of apparatus views. After running the command, indicate the first apparatus (**P1**), select the **one line symbol** and insert the symbol into the scheme. In next steps indicate other apparatus (**P2**-**P7**) and insert the received symbols into the drawing.



Switchgear view



Switchgear scheme in which some symbols have been added on the basis of switchgear view

Schematic lines – edition

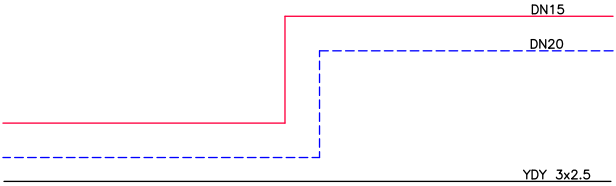
In **CADprofi** program schematic lines are drawn with lines or polylines. For lines it is possible to specify their **kind** and **type**:

Kind – means the intended use of the selected line or a specific system, such as the **supply line** in the heating installation or the **alarm line** in the monitoring installations. Schematic lines are located on layers whose names are dependent on the kind of installation.

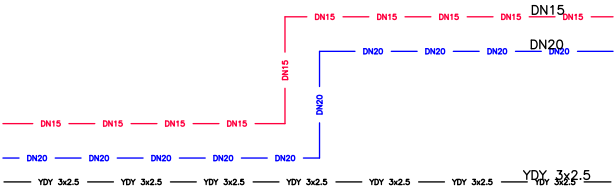
Type – determines the line size or its physical properties, for example for the **DN15** steel pipes or for **YDY 3x2,5** installation lines. Type is being stored in internal data of each line sections. Type can be edited with the **Schematic lines - edit** command.

Lines can be shown in two view types:

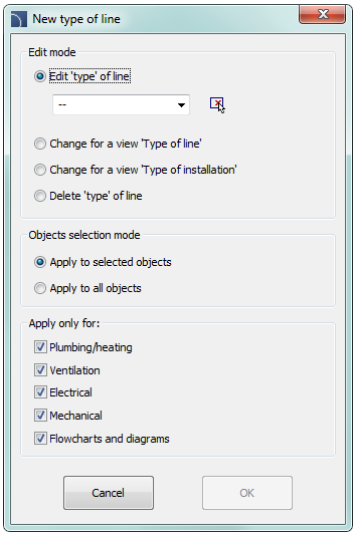
Type of installation view – shows lines in accordance with the standards or settings (in which the line kind - model was specified) that were set by the designer. This view should be used when printing projects.



Type of line view – show lines in such a way that in the lines there is a visible text that indicate the line type. **CADprofi** program allows to create complete specification of lines that were used in the project. The **Type** of line view allows to make better control when specifying line types, therefore it should be used when user adds certain types to the lines.



The **Schematic lines – edit** command allows to specify types and to change the lines view.



New type of line dialog window

The dialog window contains the following items:

Edit mode – allows to select one of available editing functions:

- **Edit type of line** – allows to specify the type, which will be added to the selected lines. This option also changes the line view into the **„Type of line"** view.
  It is possible to take the information about the line type from indicating a line that is located in the drawing with the button.
- **Change to a „Type of line" view** – changes the view in such a way that the line type is visible for each line.
- **Change to a „Type of installation" view** – changes the view in such a way that the line type is in accordance with its definition in the database.
- **Delete „Type of line" –** deletes information about the line type and restores the **Type of installation** view.

Apply to selected objects – changes the view only for indicated objects on the drawing.

Apply to all objects – changes the view only for all objects on the drawing.

Hint

Using the "Apply to selected objects" function gives the possibility to choose the lines through a selection window. If in the selection window user indicated a line that belongs to an installation that wasn't included in the dialog window then this element will be skipped.

In the "Type of line" view it is possible to quickly copy types between lines with using the CAD program command "Match properties".

Procedures

Changing the line type

1. Run the **Schematic lines – edit** command.
2. In the dialog window select one of available edition modes: **Edit type of line**, **Change to a „Type of line" view**, **Change to a „Type of installation" view** or **Delete „Type of line"**.
3. (Optional) If the **Edit type of line** have been selected, then specify the type or click the **Copy from line** button, to indicate a line in the drawing from which the information about line type will be gathered.
4. Select the **Apply to selected objects** or **Apply to all objects** option.
5. (Optional) Select installation types, which will be included during object selection.
6. Click the **OK** button to close the window.
7. (Optional) If the **Apply to selected objects** has been selected then indicate elements that you would like to edit and press the **Enter** key.
8. Selected lines will be changed accordingly to the selected setting in the dialog window.

Schematic lines – edition

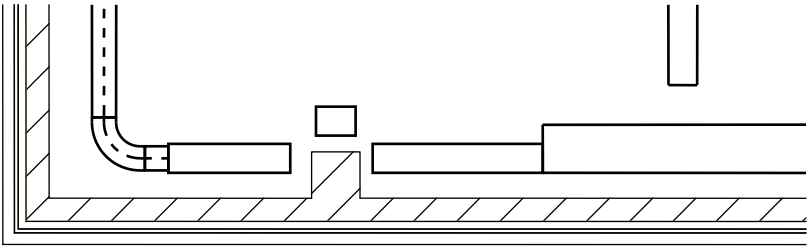
2D Join pipes/ducts

The fastest and the most convenient way to create 2D installations, such as air condition, piping or cable trays is to draw them The fastest and the most convenient way to create 2D installations, such as air condition, piping or cable trays is to draw the lines or sequences of installation lines, which are later automatically connected with the help of the universal Join pipes/ducts command. During the connection creation, **CADprofi** program lengthen or shorten connected lines and automatically inserts optimal fittings.
Connections can be made on elements that have certain types and connector size. It means that pipes, ducts, trays, armature, pumps and many other devices can be connected.
After running the **2D Join pipes/ducts** command, user should indicate lines that he would like to connect. Depending on the position and size of the indicated lines, **CADprofi** program will propose the right fitting.

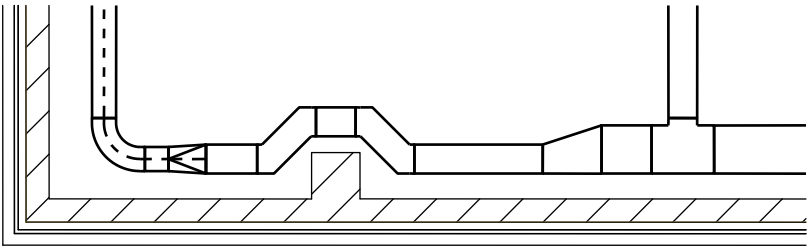
The fitting selection and the determination of its size is made in the dialog window. The dialog window layout is always the same as in commands that are used to draw lines, ducts and fittings.

Example:

- In the first stage the lines are drawn.



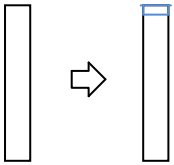
- In the second stage the lines are connected with automatic insertion of appropriate fittings.



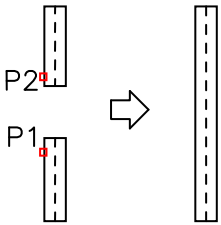
Rules that are used for connecting

- piping installations,
- air condition lines,
- cable trays.

- One line:
  Insertion of a flange, nipple, coupling and a cap with the possibility to lengthen or shorten the line.



- Two, three or four coaxial lines with the same cross-section:
  Connections without fitting.

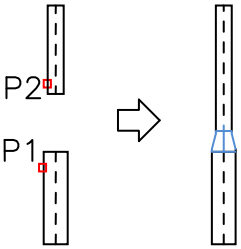


- Lines inserted at the same angle:

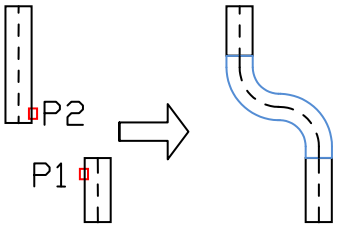
  Connections with a symmetrical reduction, asymmetrical reduction or fuller.

  For air condition it is possible to insert circular, rectangular or circular-rectangular fittings.

  Fitting is inserted in such a way as not to alter the length of the first indicated line (**P1**).



reduction

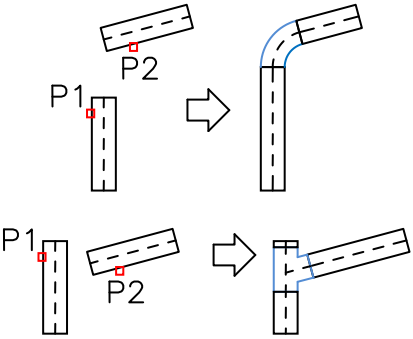


fuller

- Elements inserted at different angles:

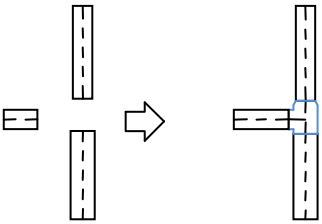
  Connection made with an elbow or a tee.

  When connecting with a tee user should first indicate the main line and after that the second line.



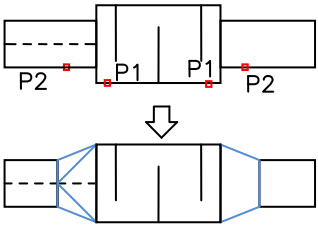
- Three lines from which two are coaxial:

  Connection with an inserted tee or a tee (reduced).



- When connecting flex lines with other lines inserted at different angles, the same rules are used as for circular lines.
- Some connections can't be made due to lack of appropriate fittings, e.g. there are not tee flex fittings.
- Connection of equipment elements of air condition ducts with other ducts:

  Connection with inserted reducer.



- It is also possible to make connections between other objects e.g. between armature, objects, lines and fittings. In each case, the rules for creating connection are the same as when connecting pipes or ducts.
- Program also allows to connect lines that are insulated.

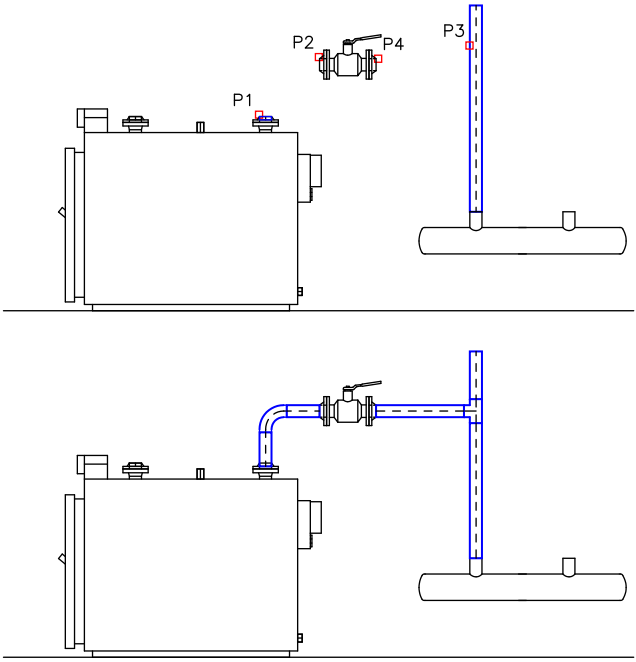
> ***Notice***
>
> ***Application doesn't allow to automatically connect lines and ducts that belong to different kinds of installation. If it necessary to connect for example a piping installation with an air condition installation, user will have to insert appropriate fittings manually.***

Procedures

Connecting lines

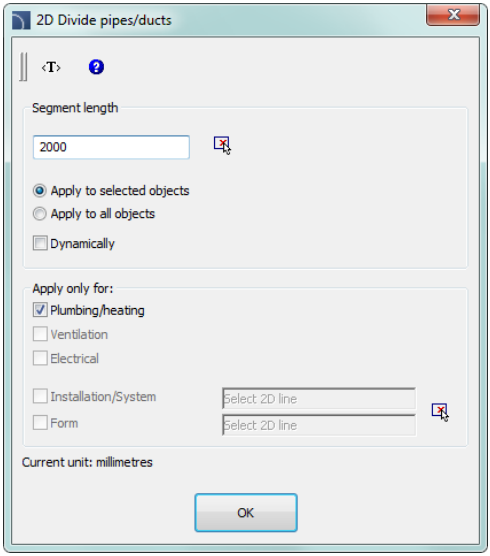
1. Run the **2D Join pipes/ducts** command.
2. Indicate elements that you would like to connect. Confirm by clicking the right mouse button or by pressing the **Enter** key.
3. In the newly opened dialog window select the fitting type that you would like to use.
4. (Optional) Specify the necessary dimensions of selected fitting.
5. (Optional) Specify the **Thickness of insulation** value for the selected fitting.
6. Press the **OK** button to close the window and make the connection.

Examples of connections between fittings, lines and objects:



2D Divide pipes/ducts

The **2D Divide pipes/ducts** command allows to divide 2D lines into segments. During the initial design stage, the easiest way to draw installation is to not pay any particular attention into individual segment length. In the final design stage it is possible to divide long lines into separate segments, which will be later ordered during the implementation of the project. It is possible to quickly divide all ducts into segments that have desired length or a dynamic division of single lines.



Divide pipes dialog window

The dialog window contains the following items:

Segment length – specifies the segments length, on which lines are being divided. It is possible to specify the length (by indicating two points) from the drawing after clicking the **Specify on screen** icon.

Apply to selected objects – divide into segments only lines that are indicated in the drawing.

Apply to all objects – divide into segments all lines from the chosen installation type.

Dynamically – enable/disable the dynamic division mode. The dynamic mode allows to divide lines into segments that have different lengths. The segments lengths can be indicated by entering a value in the command line or by indicating points.

Apply only for – the selection of installation kind/type, on which the line division will be made. This options work with both the **Apply to selected objects** giving user the opportunity to select lines from the drawing that belong to a certain installation

kind and with the **Apply to all objects** option that allows user to divide all lines from the selected kind of installation.

> ***Hint***
>
> ***Using the "Apply to selected objects" function gives the possibility to choose the lines through a selection window. If in the selection window user indicated a line that belongs to an installation that wasn't included in the dialog window then this element will be skipped.***

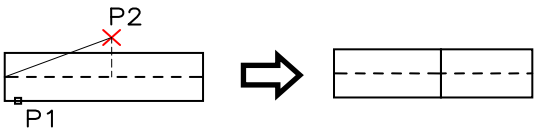
Installation/System –allows to select an appropriate system, which lines will be divided, for example **Supply**, **Return** etc. This options work with both the **Apply to selected objects** giving user the opportunity to select lines from the drawing that belong to a certain installation kind and with the **Apply to all objects** option that allows user to divide all lines from the selected kind of installation. Installation kind can be selected by pressing the **Select 2D line** button.

Form – this option allows to specify the form of lines, on which the division will be made for example circular, rectangular etc. in order to determine the data click the **Select 2D line** button and then select a line from which you would like to gather data.

Procedures

Dynamical line division

1. Run the **2D Divide pipes/ducts** command.
2. In the **Divide pipes** dialog window enable the **Dynamically** option and then click the **OK** button. After the dialog window will close, select the appropriate line (**P1**).
3. Application will automatically determine the closest end point, from which you can start dividing the line into segments.
4. In the command line enter the new segment value. This value will be measured from the starting point and will divide the line.
5. (Optional) Indicate a point that on the line (**P2**) in order to specify the new segment length.
6. (Optional) Repeat step **4** or **5** to continue the process of dividing a line.
7. End the command with the right mouse button or the **Enter** key.

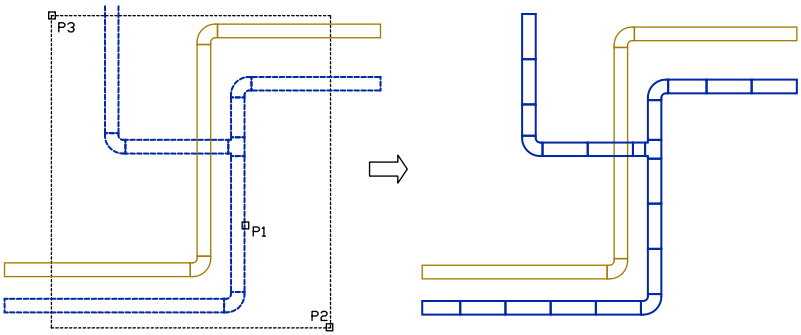


Dynamic line division

Procedures

Division of many lines into segments

1. Run the **2D Divide pipes/ducts** command.
2. In the **Divide pipes** dialog window disable the **Dynamically** option.
3. Specify the length of segments, on which the lines or ducts will be divided.
4. Enable the **Apply to selected objects** or **Apply to all objects** option.
5. Select the installation kind or with the help of the button indicate the needed element to specify the system and the form of divided lines or ducts.
6. After clicking the **OK** button the dialog window will close.
7. If the **Apply to selected objects** was enabled, then the program will allow to select elements that will be divided (P2, P3).
8. End the command with the right mouse button or the **Enter** key.



Division into segments of one system (the lines from the second system were not divided)

Notice

Dividing lines may also work for lines that are not visible on the screen. With a large number of lines it may take a long time for the operation to complete, therefore in the command line user is able to see the operation status.

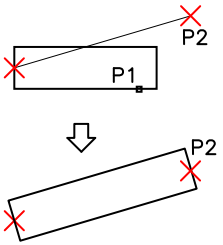
2D Edit and info

The **2D Edit and Info** command allows to edit 2D lines and fittings. This command depends on the type of the selected object. If user directly indicated any line segment, then it will be possible to edit it length by indicating a new end point. If user selects the **Edit** ("E" key) in the command line then a dialog window will appear allowing user to edit the selected segment parameters. Indicating any fitting will open the dialog box that allows user to edit all fitting parameters.

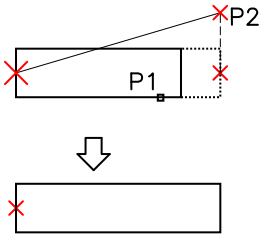
Procedures

Line edition

1. Run the **2D Edit and Info** command.
2. Select the line that you would like to edit.
3. Specify the ending point for the selected line.
4. (Optional) Select the **Change length** („L" key in the command line) option to specify a new length without changing the line insertion angle. Optionally it is possible to select the **Delta** option to specify the length of the line lengthen (in drawing units) or the **Percent** option for lengthening the line in percent.
5. (Optional) Select the **Edit** option from the context menu or from the command line ("E" key) in order to open the dialog window that gives user the possibility to change line parameters.



Direct change of the line length



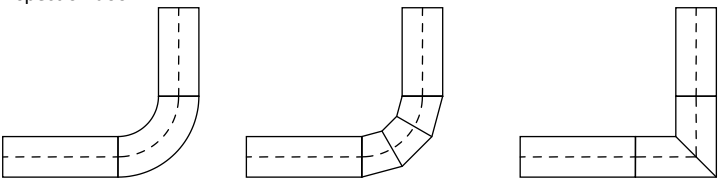
Edition after selecting the Length option

Procedures

Fitting edition

1. Run the **2D Edit and Info** command.
2. Select the fitting that you would like to edit.
3. In the dialog window specify new parameters for the selected fitting.
4. Click the **OK** button to update the fitting on the drawing.

For some fittings it is possible to change their type, for example when editing elbows it is possible to change the elbow into a segmented, two-segmented or to an elbow with an inspection door.



Examples of changing the type of fitting

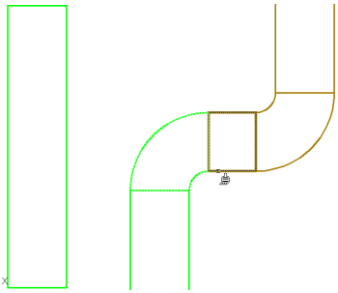
When editing fittings user can use the **Copy dimensions >** option, thank to which in an easy and convenient way he can get dimensions from other elements that are located in the drawing.

Changing the system type for lines and 2D fittings

Elements that belong to the appropriate system or installation are automatically located on specified layer. The **2D Edit and Info** command allows to change the system for individual fittings. System change can also be performed by moving elements into another layer. In CAD programs there are many ways to change object layers. A very convenient way is to use the match properties command:

Example:

To change the system (for example in air condition installation from supply air into exhaust air), user should run the **Match properties** command, select the source object and then select all objects for which he would like to change the layer into exhaust air installation layer.



Notice:

Circular, rectangular and flex lines that belong to the same system are located on different layers:

– circular lines are located on the 2Dx_**PRO**_xxx layer

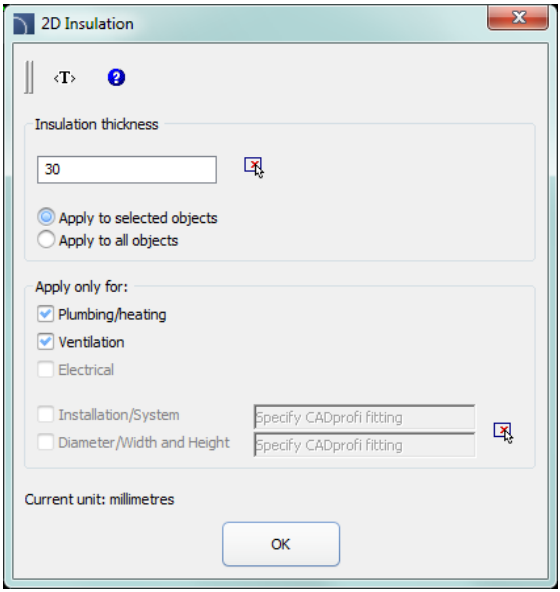
– rectangular lines are located on the 2Dx_**PRE**_xxx layer

– flex lines are located on the 2Dx_**PFX**_xxx layer

The beginning of layer name (2DL) means air condition, the next part (PRO, PRE or PFX) means line form, and the ending (xxx) means system or installation.

2D Insulation

The **2D Insulation** command allows to insulate lines. It is possible to specify the thickness for insulation and to delete insulation.



Edit insulation dialog window

The dialog window contains the following items:

Insulation thickness – determines the insulation thickness, which will be added to selected lines. It is possible to determine the thickness by indicating two points from the drawing (**Copy from fitting >** option). To delete insulation from lines, user has to specify the insulation thickness as "0". For air condition lines it is also possible to specify their internal insulation by entering a negative insulation thickness.

Apply to selected objects – edit insulation thickness only for lines that are indicated in the drawing.

Apply to all objects – edit insulation thickness for all lines from the chosen installation type.

Apply only for – the selection of installation kind/type, on which the edition of insulation thickness will be made. This options work with both the **Apply to selected objects** giving user the opportunity to select lines from the drawing that belong to a certain installation kind and with the **Apply to all objects** option that allows user to specify the insulation thickness for all lines from the selected kind of installation.

> ***Hint***
>
> ***Using the "Apply to selected objects" function gives the possibility to choose the lines through a selection window. If in the selection window user indicated a line that belongs to an installation that wasn't included in the dialog window then this element will be skipped.***

Installation/System – allows to select an appropriate system, which lines will be divided, for example **Supply**, **Return** etc. Kind of installation can be chosen by pressing the **Specify CADprofi fitting** button.

Diameter/Width and Height – allows to specify line dimensions, for selected lines, e.g. D=114.3 for DN100 pipes. In order to specify the data for this option, user has to click the **Specify CADprofi fitting**, and then select line, from which you would like to get data.

Procedures

Insulating lines or editing insulation thickness

1. Run the **2D Insulation** command.
2. In the dialog window specify the insulation thickness. The "0" value removes insulation.
3. Enable the **Apply to selected objects** or **Apply to all objects** option.
4. (Optional) Select types of installation, which should be included during object selection.
5. (Optional) Indicate a line from the drawing in order to specify the **Installation/System** option and the **Diameter/Width and Height** option. If it's necessary you can disable one of these options.
6. Click the **OK** button to close the window.
7. (Optional) If you have enabled the **Apply to selected objects** option indicate elements that you would like to edit and press the **Enter** key.
8. Selected objects will be insulated.

Insulated lines

Quick tools

CADprofi program has got additional commands which make it easier to create technical drawings. These commands are available in both upper menu and in the **Quick tools** toolbar.

In the current program version there are several command available including commands that make it easier to create construction lines, axes etc.:

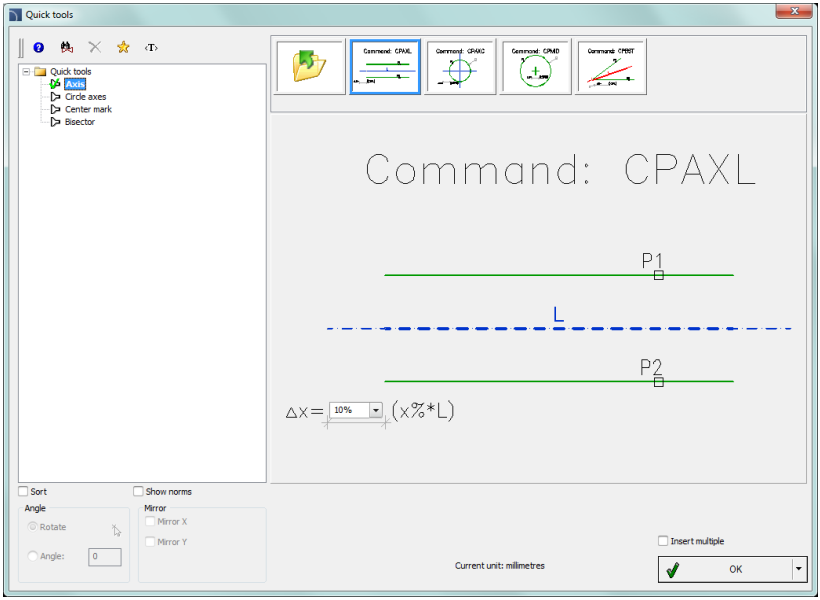
Quick tools this command displays a dialog window with quick tools. In this dialog window it is possible to specify the default lengthen of created axes, bisectors and center marks. Lengthening can be determined in drawing units or in percent.

Axis Creation of an axis after user indicates two lines.

Bisector Creation of a bisector between two lines.

Circle axes Quick insertion of axes for circles.

Center mark Insertion of a center mark for a center or a an arc.



Quick tools dialog window

Depending on the selected command, user will be requested to indicate a circle, an arc or a line.

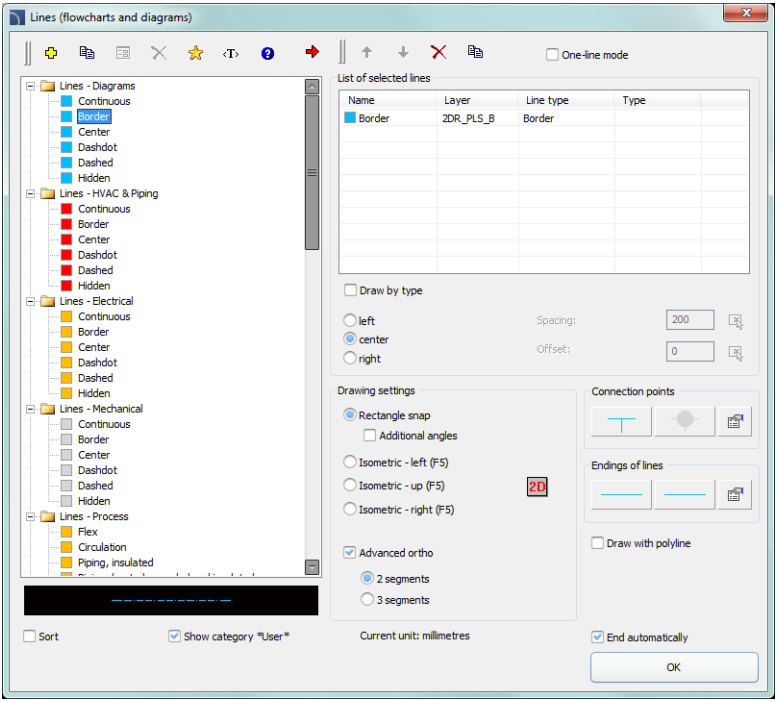
Quick tools

Flowcharts and diagrams

CADprofi allows to create logical, functional and technological block diagrams. A large library of symbols allows to use these diagrams in all program modules.

Diagrams lines

The **Lines (flowcharts and diagrams)** command allows to draw block diagrams lines in a convenient way.

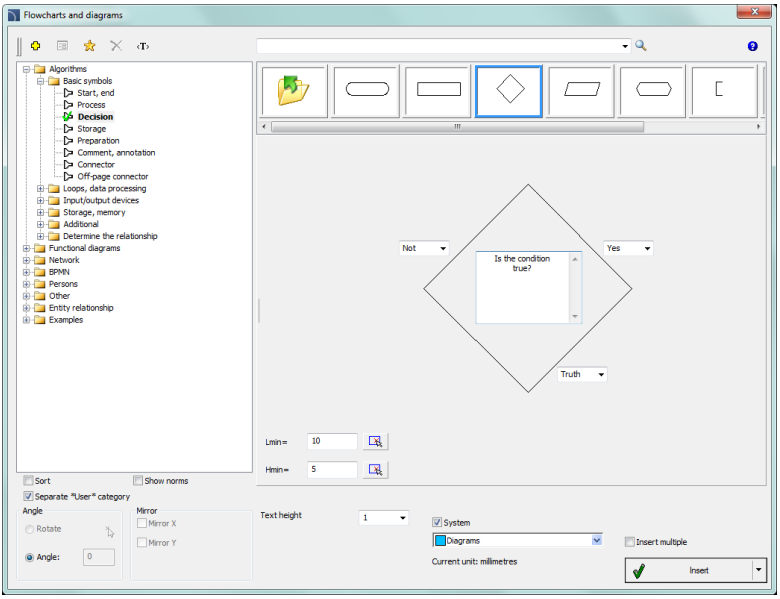


Lines (flowcharts and diagrams) dialog window

How to draw lines is fully explained in the individual modules chapters.

Flowcharts, block diagrams

The **Flowcharts and diagrams** command contains elements that are used to draw a variety of functional diagrams, network infrastructure, technological process models and block diagrams.

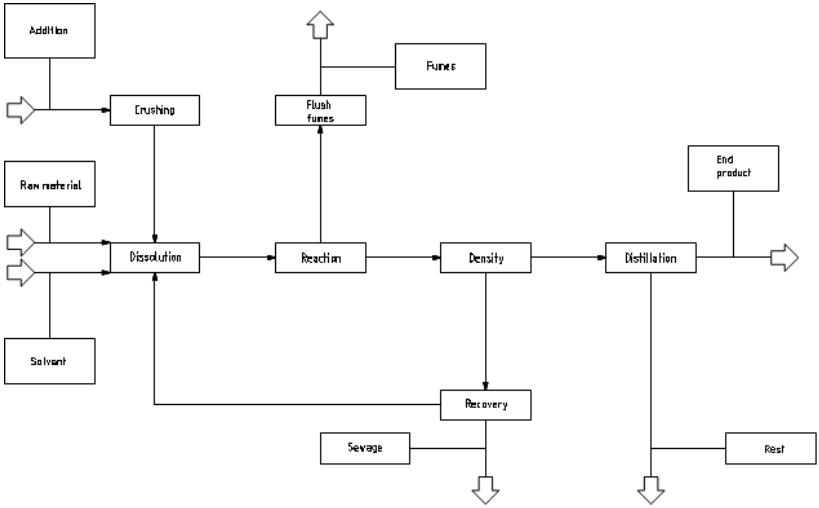


Flowcharts and diagrams dialog window

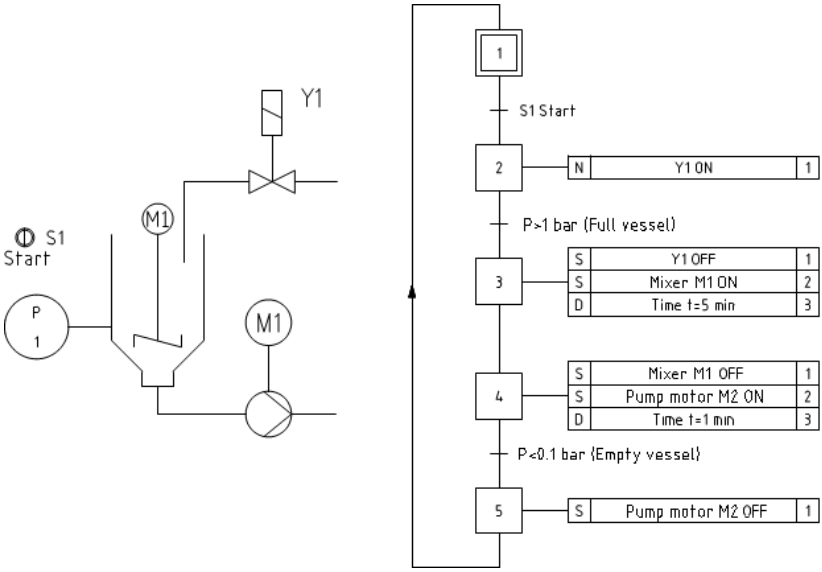
Many block diagrams elements has got edit fields, in which it is possible to enter a variety of texts. Program automatically adjusts the size of elements that are inserted into the drawing depending on the text length. If user needs to edit the text then he can use the **Edit symbols** command, thanks to which after making the edition the block size will be updated. It is also possible to determine the minimum size of the block, so even when adding small amount of texts the size of individual elements will be similar.

All ways of working with symbols are fully explained in the individual modules chapters.

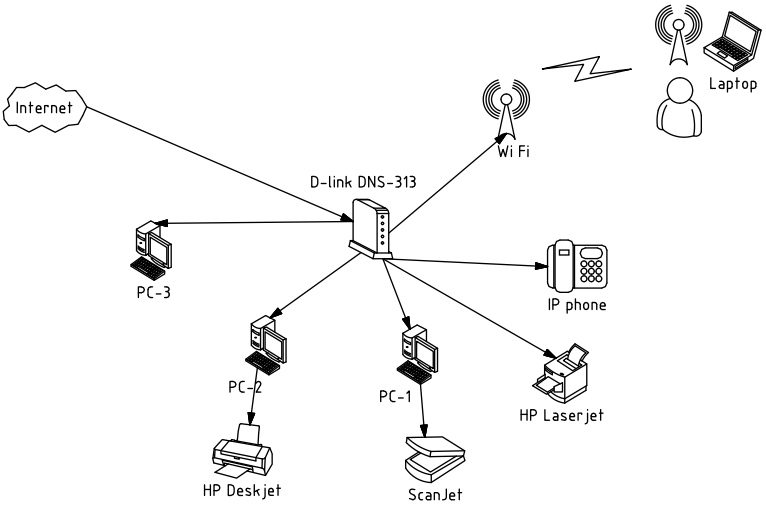
Examples of block diagrams and flowcharts:



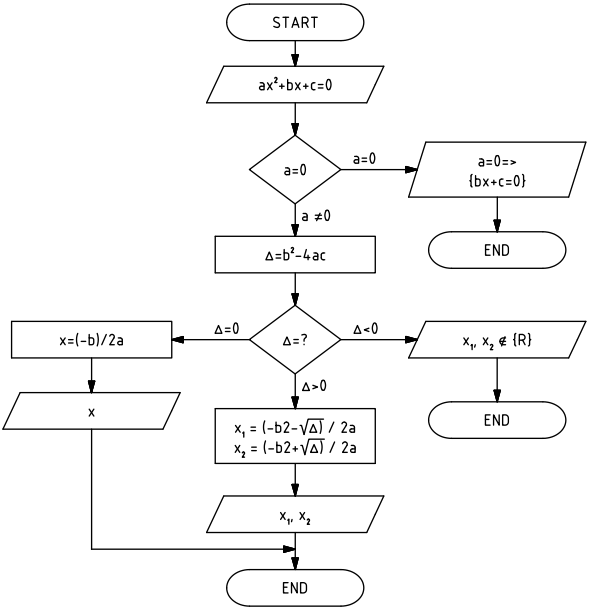
This drawing shows an example block diagram of an industrial installation.



Functional diagrams, drawn according to the EN 60848 standard.



Creating computer network and infrastructure diagrams.



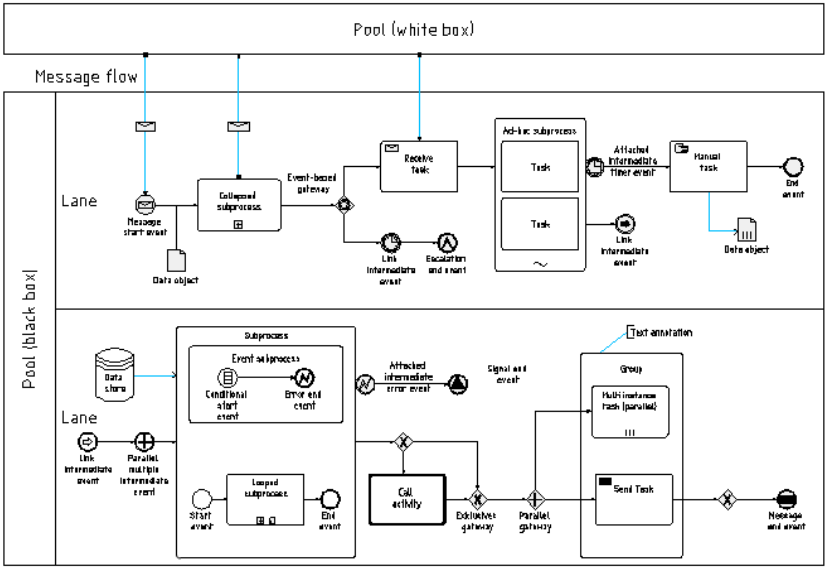
An example of an algorithm

Program can draw BPMN diagrams. Business Process Modeling Notation is a graphical way of showing business processes.
BPMN describes three basic types of processes:

- internal process
- public process
- cooperation process

In **CADprofi** program the following BPMN graphical elements categories are available:

- active elements
- connections
- location of the process
- graphical elements used to add additional information (data, annotations and groups)



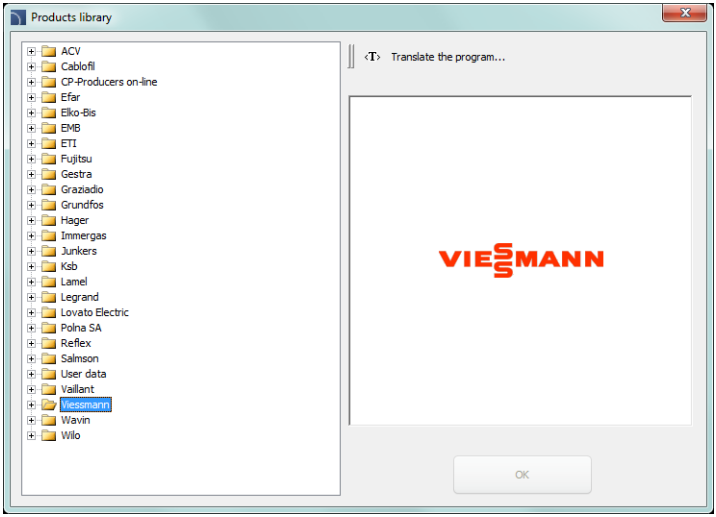
An example of a BPMN diagram

Manufacturers library

Manufacturers' libraries contain blocks of many machinery and construction equipment producers. Objects are shown in different views. Many objects are also available as 3D models. Thanks to the parametrical technology, individual libraries take up very little space. This technology also allows to automate many tasks that are involved with designing or preparing specifications.

The **Products library** command allows to select objects and symbols that are defined by user that used the **Add objects to library** command.

Libraries are also distributed by the manufacturers that have got a license for the **CP-Manufacturers** program. Because of that it is possible to expand manufacturer libraries at any time by installing additional libraries that are provided by the manufacturers.
The **Products library** command allows to select the active library. After inserting elements from the selected library it is possible to open the same library again by clicking the **Previous library** command. Thanks to this user omits the manufacturer selection window.



Manufacturer selection dialog window

After selecting the library a dialog window will appear in which user can view the selected devices or schemes for the currently selected manufacturer. Library structure may be different, depending on the selected manufacturer, but the way how to work with manufacturers objects is very similar to working with standard **CADprofi** objects, that were described in the **CADprofi** modules sections.

Extending program content

Program has got an open structure, thanks to which user can add new elements and expand the databases that are available in the program on his own. It is possible to add to the program user created lines, symbols, objects, parametrical devices, tables etc. When user wants to expand the program content it is very useful for him to use the **Define new** command, thanks to which he will be able to automatically add blocks to the objects or symbols folder.

Extending program databases

In many dialog windows there are buttons that allow to expand and edit the program databases:

Define new – allows to add a new element to the database
New element is usually added to the ***User*** category.

Copy – Copies the selected element to the ***User*** category.

Edit – allows to change selected elements.

Delete – deletes selected elements from the user database.

Add to favourites – creates the ***Favourites*** subcategory in which user can add elements that he is using the most often.

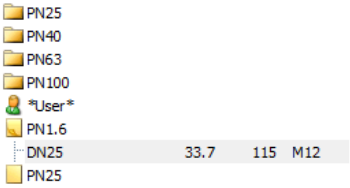
> ***Notice***
>
> ***It is possible to delete and edit only these elements that were created by the user.***

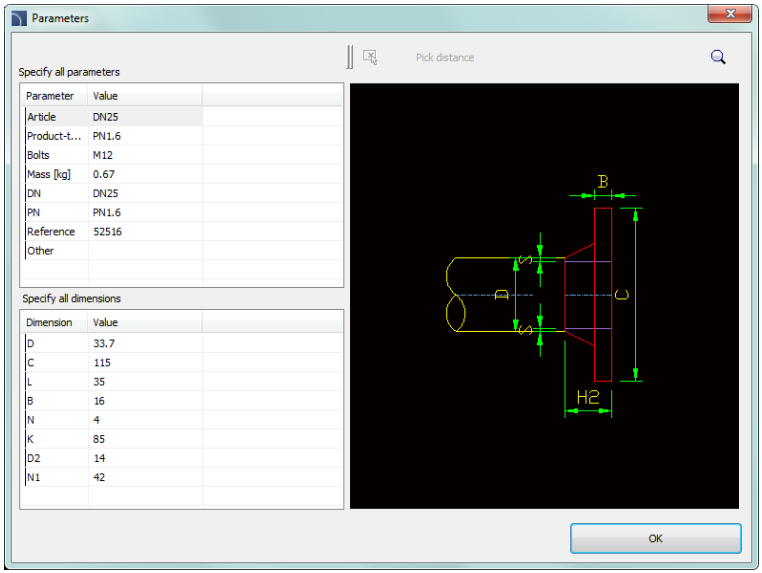
Data that is added by the user is stored in the *UserData* folder. This folder is located in the main program folder (e.g. *C:\Cadprofi*).
Individual data is saved in the **.uuc* files. These files have got a similar structure to the **.csv* files. For this reason advanced users can edit them in a spreadsheet or in a text editor, for example in the Notepad. Less advanced users should only expand databases by using **CADprofi** appropriate commands.
To save changes that were made in the user files, the *UserData* folder should be archived. Thanks to this it will be possible to quickly restore the data after user decides to reinstall **CADprofi** software for example on another computer.
When **CADprofi** program is being updated, user data is overwritten, therefore they can be used in any new **CADprofi** program version.

Creating and extending user type of series

Program contains many object databases with specified parameters. User has the possibility to extend the database with his own type of series, which can be freely edited and saved in the *UserData* folder.

User types of series are created on the basis of copies of existing library elements. All copied positions are visible in the ***User*** category. Renaming the type of series in its new position will move it to the appropriate folder.





Parameters dialog window

Procedures

Creating and extending the user type of series

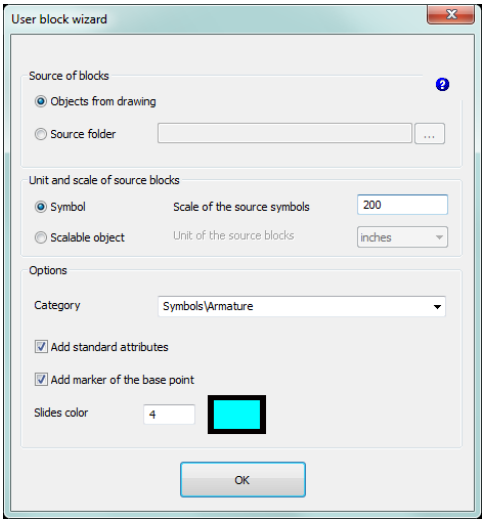
1. In the dialog window that contains the parametrical objects click and select the element, on which basis the user type of series will be created.
2. Select the **Copy** command that is located at the top of the dialog window. Selected element will be copied and placed in the *User* category.
3. (Optional) Select the **Edit** command to change the dimensions and attributes for the new type of series position.
4. Insert object or close the dialog window.

Defining user blocks

Each library in **CADprofi** consists of CAD blocks and the database in which names and other parameters are defined. Blocks are saved in the **.dwg* files, which have an appropriate structure. When adding new blocks, **.sld* slide files are also created. In these files the miniatures (preview) are located, which are normally displayed in dialog windows.

In older version of the **CADprofi** program in order to automate the process of adding blocks the **User block wizard** command was being used. All elements used in previous versions of **CADprofi** program are still available and are now located in the **Products liblary** command

Users in the latest version have got the possibility to expand the program library directly in the dialog windows that contain symbols. The **Define new** button is used for that. It runs the User blocks wizard. It is possible to define blocks on the basis of elements from the drawing or by importing already made **.dwg* files, which are saved in the selected source folder. All data that is created during the defining of user blocks is saved in appropriate folders in *C:\CADprofi\UserData*.



User block wizard dialog window

The **User block wizard** dialog window contains the following elements:

Source of blocks – options that allow to specify how to define new blocks:

Objects from drawing – creates a block from the objects indicated from the current drawing.

Source folder – creates blocks from all **.dwg* and **.dxf* files that are saved in the indicated patch with the exception of any possible subfolders.

Unit and scale of source blocks – allows to select the type of object and how to scale it during the insertion to the drawing. If the source blocks are drawn in scale, then the parameters should be set in such a way that blocks will be scaled to **meters**.

Symbol – block scaled uniformly, requires the determination of source symbol scale.

Example: Symbol on the drawing has got a 200 length.
In order to have the target symbol with a 1 length, the source block should be reduced 200 times. Therefore the provided scale should be:
1 / 200 = 0.005

Scalable object – block that is scaled arbitrarily in X, Y or Z directions, requires to specify the unit for the source block.

Example: The real device dimensions are 20 cm x 30 cm.
The block that shows this device has got 200 x 300 dimensions.
It means that this block was drawn in mm. Therefore in the dialog window user should select the mm as a unit and scale as 1:1

Category – allows to specify the category, where new blocks will be added.

Add standard attributes enables/disables the addition of standard attributes to the created block.

Add marker of the base point – enables/disables the visibility of the base point in the preview.

Slides color – allows to select the slide color of created blocks. Slide color is set only for these blocks elements, which were drawn on the **0**" layer and with the **ByLayer** color.

After pressing the **OK** button the program begins the operation. If the blocks are added from the source folder, then blocks will be loaded into the drawing consecutively from this folder. These blocks are appropriately modified and saved in the library.

Procedures

Adding blocks as an object fromt he drawing

1. Make sure that in the drawing, there is an element that will be converted to a block.
2. In the dialog window that contains symbols click and select the position on the basis of which the parameters of the new block will be created.
3. Click the **Define new** command that is located in the upper parts of the dialog window, afterwards the **User block wizard** will be opened.
4. In the **Source of blocks** section select the **Objects from drawing** option.
5. (Optional) Change the settings in the next dialog window sections in accordance to your own preferences.

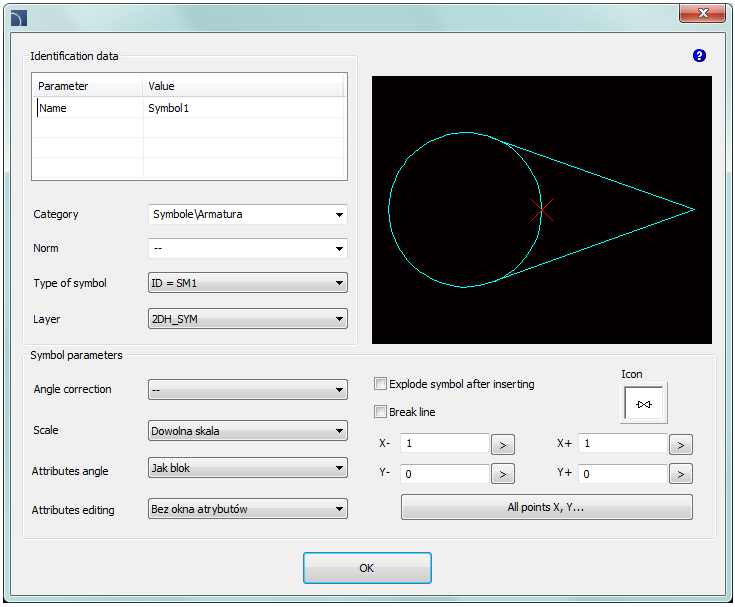
6. Click the **OK** button, afterwards the dialog window will be closed.
7. Select elements, that you want to change into a block, confirm by clicking the **Enter** key.
8. Select the (insertion) base point for the created block.
9. Program will perform all the operations that are necessary to add an item to the database.

Adding blocks from the soruce folder

1. Create a **New drawing** or make sure that the currently used is empty.
2. In the dialog window that contains symbols click and select the position on the basis of which the parameters of the new block will be created.
3. Click the **Define new** command that is located in the upper parts of the dialog window, afterwards the **User block wizard** will be opened.
4. In the **Source of blocks** section select the **Source folder** option, click the browse ... button and indicate the folder in which the **.dwg* and/or **.dxf* files that you would like to add to the **CADprofi** library are located.
5. (Optional) Change the settings in the next dialog window sections in accordance to your own preferences.
6. Click the **OK** button, the dialog window will be closed and the application willl automatically open each of the indicated location adding it to the **CADprofi** library. Insertion point in this case will be the 0,0,0 point from each of the drawing.

Editing user block parameters

Program allows to edit both the basic identification data, such as: name, category, norm, type of symbol or layer, as well as more advanced symbol parameters.
In order to start the data editor window user should select the **Edit** command that is located in the symbols library menu.



In the dialog window user can specify the following elements:

Name – allows to select the name which will be shown in the dialog window and in BOMs. Name can be changed after pressing the **F2** key directly in the library window.

Category – allows to select the category, in which the selected block will be placed. It is possible to enter any category name in the **Category\Subcategory** system.

Norm – standard selection, to which the block will be assigned. It is possible to define a new standard by the User.

Type of symbol – internal program settings that contain information about the block parameters, for example the block insertion angle.

Symbol parameters –settings that describe the behavior of the block during insertion.

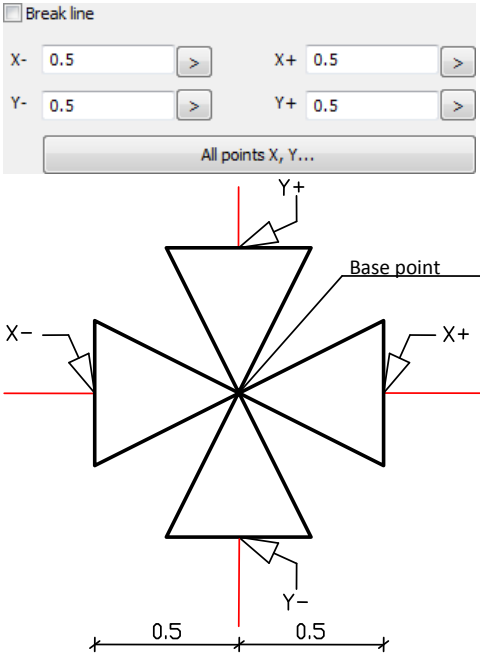
Angle correction – settings for a permanent or any block rotation angle.

Scale – settings for a permanent or any block scale.

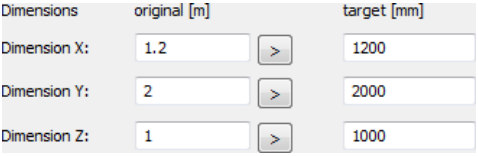
Attributes angle – settings for the attributes in the block rotation angle.

Attributes editing – settings for block attributes edition.

Break line – enables/disables the possibility to insert the symbol into lines with automatic interruption in the connection points. Conenction points are specified in the **X–**, **X+** and **Y-**, **Y+**fields. In these fields user should enter the actual length of the cut in a given direction from the insertion point for a symbol in a 1:1 scale. These values can also be indicated in the drawing after clicking the > button or after selecting the **All points X, Y… >**. The vast majority of the symbols uses the intersection points only in the **X** axis. One exception being the **Cross symbols**, which are inserted at the intersection of two lines.



Dimensions – option that allows to scale the selected symbol independently in three main directions (**X**, **Y**, **Z**). This option is only available for blocks created as a **Scalable object**.



CADprofi software translation tutorial

What is this chapter about?

CADprofi program possesses a very rich content, including a large number of national and international standards, manufacturers' libraries etc. Many standards or libraries are introduced to the program only in their original language or are translated only to some languages. User can add his own translations, thanks to which it will be possible to obtain BOMs and to easily create description in user native language.
It is also possible to share translations to other users and to send them to CADprofi, thanks to which they will be included in next program versions.

This chapter describes how to translate CADprofi interface, module content (**Mechanical**, **Electrical**, **HVAC & Piping**, **Architectural**) and manufacturers' libraries.

Requirements

In order to prepare user own translations it is required to have CADprofi **9.21 (or later version)**. It is also recommended to prepare translations in a Unicode software such as **AutoCAD 2007 (or later)**, **BricsCAD V9 (or later)**.

Translations can be also prepared in exported PO format files. In order to edit these files, user should install the POEDIT software (freeware), which can be downloaded directly from the: *http://www.poedit.net* website.

The localization process cycle

The following drawing shows the **CADprofi** localization process cycle.

CADprofi context translation – translation of interface and program content through the tools that are built in the **CADprofi** program. User can add translations on their own need or comprehensively translate the whole program.

Verify – verify/translation in external files through the **POEDIT** software. User can use this method to cooperate with external translation office or agencies.

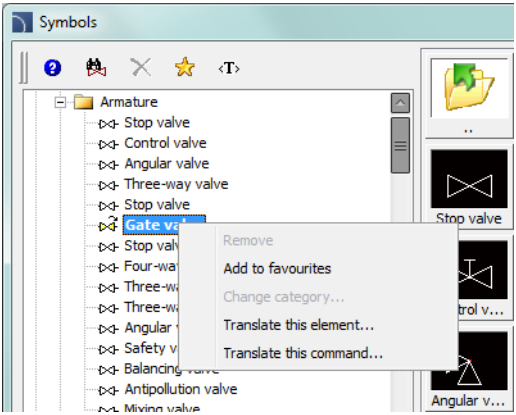
Send to CADprofi – every user can send his translations as his translation suggestions. After verification, they will be added to the program in next **CADprofi** updates.

CADprofi context translation

This method makes it possible to translate by adding new text or by replacing an existing translation directly in the **CADprofi** program.
User translations replace the standards program translations.
All translations made with this method are saved in the '**UserData**' folder that is located in the **CADprofi** main folder (default: *C:\CadProfi*).

Translating the current command

It is possible to translate the current dialog window (options, settings) and the items contained in this window. This is a convenient method, because user translates what he sees at the moment on the screen.
Translation is made by using the dialog window that contains a translation table. This window can be invoked from the context menu by pressing the **Translate element** or **Translate command** option which is visible after right clicking on the mouse button on the selected block.



Translate element – enables the translation of selected symbol/object name and its additional information connected with this element (e.g. standard name).

Translate command – enables the translation of the interface of the currently opened dialog window.

After using the **Translate symbol/ Translate interface** command a dialog window will appear allowing to translate texts. This window contains the following items:

Name/No – describes a value like name, norm, parameter, etc.

Source – source text

Standard translation – original translation in the currently selected language. This value will be empty if it's untranslated in the current language.

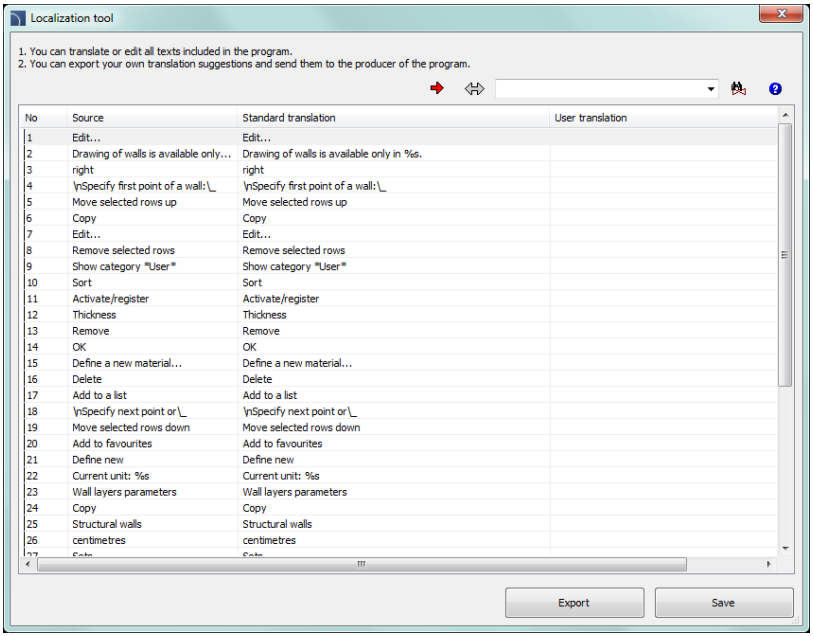
User translation – editable field that allows to add user translations.

Get source text ( ) – copies the selected source text to the **User translation** column.

Get standard translation ( ) copies the standard translation to the **User translation** column.

Search ( ) – searches for the entered phrase in all three table.

Export – allows to export translations to exterior **PO** files for verification and optional sending to **CADprofi**.


Localization tool
1. You can translate or edit all texts included in the program.
2. You can export your own translation suggestions and send them to the producer of the program.
No
Source
Standard translation
User translation
1 Edit... Edit...
2 Drawing of walls is available only... Drawing of walls is available only in %s.
3 right right
4 \nSpecify first point of a wall:_ \nSpecify first point of a wall:_
5 Move selected rows up Move selected rows up
6 Copy Copy
7 Edit... Edit...
8 Remove selected rows Remove selected rows
9 Show category *User* Show category *User*
10 Sort Sort
11 Activate/register Activate/register
12 Thickness Thickness
13 Remove Remove
14 OK OK
15 Define a new material... Define a new material...
16 Delete Delete
17 Add to a list Add to a list
18 \nSpecify next point or_ \nSpecify next point or_
19 Move selected rows down Move selected rows down
20 Add to favourites Add to favourites
21 Define new Define new
22 Current unit: %s Current unit: %s
23 Wall layers parameters Wall layers parameters
24 Copy Copy
25 Structural walls Structural walls
26 centimetres centimetres
Export
Save

Context translation dialog window

In case of translating the command in the table only the texts that have been used by program during the command execution will be available. Because of that it is recommended to use all options that are available in the command before pressing the **Translate command** option. Thanks to this the translation system will record all text and will make them available for editing. It should be noted that some texts associated with the command are only available after user closes the dialog window (e.g. after insertion of selected symbols into the drawing). These texts will be also recorded and made available for editing, but only when user runs the command once again.

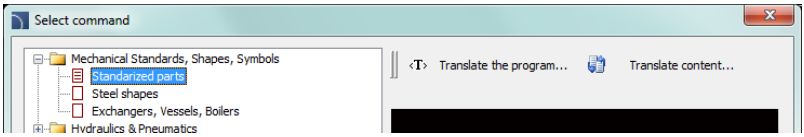
Procedures

Context translation

6. Open any **CADprofi** dialog window.
7. Right click on a symbol which name you want to translate.
8. Choose the **Translate element/Translate command** to run the translation window.
9. (Optionally) Type in the command line CP_TRANSLATION to run the translation window of the last used **CADprofi** command.
10. Change the texts that are in the **User translation** column.
11. Click the **Save** button to close and save changes.
12. All changes will be made after you reopen the **CADprofi** dialog window.

Translating the whole program

When translating contextual translations, the system record all texts that are used by the program and make them available for translation. In this way it is possible to translate all texts, which are visible when user "normally" uses the program. Apart from standard texts, CADprofi program also contains many additional messages that appear sporadically (such as messages about the improper program operation). The best way to translate these texts is display a full table of texts in which user can easily find the text that requires translation. The collection of all texts that are used in the program as well as the names of all symbols and objects can be displayed with the **Translate the program** and **Translate content** that are available in the modules main command: **Architectural**, **HVAC & Piping**, **Electrical**, **Mechanical** and **Products library**.



Notice

Program text interfaces are the same for all modules but each module and each manufacturer's library have got their own dictionary with their content translations.

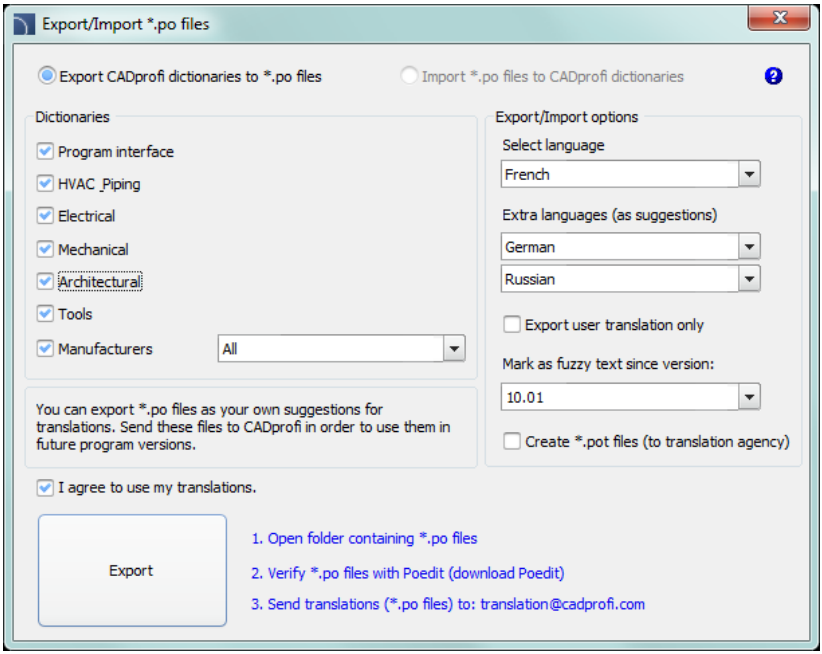
Export and translation verification

After translating the user interface, and the program content it is possible to export translations to files in **PO** format and send them to **CADprofi**, as suggested translations, thanks to which they will be available in future program updates.

> *Tip*
>
> *If you want to know more about the PO file format please visit the GNU ORG website.*

Export to PO files

To export translations to **PO** files, user should press the **Export** button in the context translation dialog window.



PO files generator dialog (export view)

The dialog window contains the following items:

Mode options – enables to choose between the **Export** and **Import** options.

Dictionaries – enables to select dictionaries to **Export** or **Import**.

Manufactures list – list of manufacturers dictionaries.

Main language – allows to select the translation language.

Extra languages – enables to choose two additional languages which translations will be visible as suggestions during the work in the **POEDIT** program.

User translation only – options that allows to generate **PO** files that will contain only data that was entered by the user during the context translation.

Mark as fuzzy text since version – allows to mark texts added from selected version number as 'fuzzy'. Thanks to this the person who prepares translations can quickly find new texts that were introduced to **CADprofi** program since a particular version.

Generate template (*.pot) – allows to generate template files (***.pot** files) which are needed to update user previous **PO** files in **POEDIT** program.

Export – creation of **PO** files.

Open folder containing *.po files – Opens folder that contains **PO** files (by default *C:\CADprofi\X.x\Localization*).

Verify *.po files with Poedit – allows to download and install the **POEDIT** program, which is necessary to edit and verify **PO** files.

Send translations (*.po files) – creates a message using user default email program. **PO** files (which will be treated as suggestions when creating new program version) should be added to this message. **CADprofi** reserves the right to review and edit the submitted translation.

> ***Notice***
>
> ***Agreeing for the use of translations is required in order to perform a PO files export.***

▼ *Procedures*

Generating PO files

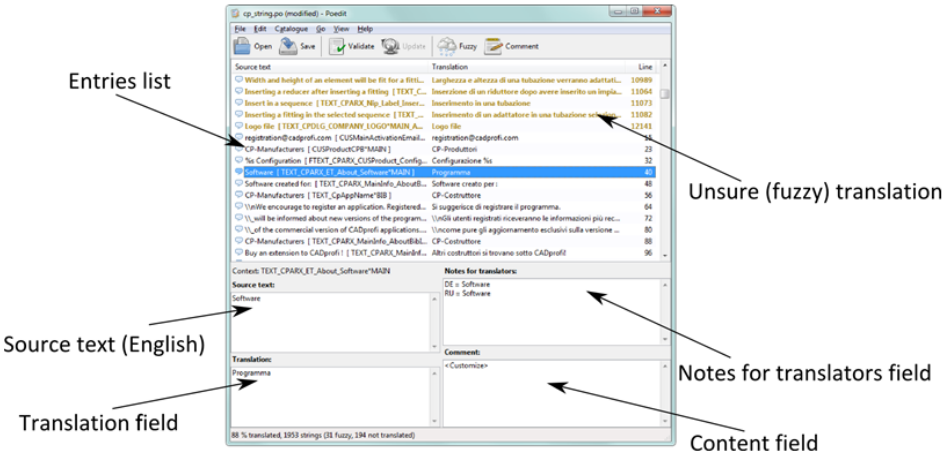
1. In the context translation dialog window press the **Export** button. The **Export/Import *.po files** window will be opened.
2. Check/uncheck dictionaries that you want to export.
3. In the **Main language** section choose language that you would like to export.
4. (Optional) Choose additional languages.
5. (Optional) Select version number in order to check translations as "fuzzy".
6. Agree to use your translations.
7. Click the **Export** button to close the window and create the **PO** files. Files will be saved in the *C:\CADprofi\X.x\Localization* folder.

User translation export

1. In the context translation dialog window press the **Export** button. The **Export/Import *.po files** window will be opened.
2. Select the **Export CADprofi dictionaries to *.po files** option.
3. Check/uncheck dictionaries that you want to export.
4. In the **Main language** section choose language that you would like to export.
5. Check the **User translation only** option.
6. Click the **Export** button to close the window and create the **PO** files. Files will be saved in the *C:\CADprofi\X.x\Localization* folder. These files can be send to **CADprofi** as translation suggestions.

Translation verification in the POEDIT program

POEDIT is a free, easy to use software that supports translations of **PO** files that were exported from **CADprofi**.



POEDIT dialog window

The **POEDIT** dialog window contains the following elements:

Entries list – list of the translated (black), untranslated (dark blue), unsure 'fuzzy' (gold) and errors (red) entries of the current ***.po** file.

Source text filed – not editable field with a source text.

Translation filed – editable field with a translated text.

Context field – information about the context of selected entry (where it is located within the program, text type etc.).

Notes for translators field – texts in additional languages, which were exported from the CADprofi program. Translations in other languages may suggest in some cases the correct translation of selected words or phrases.

> *Tip*
>
> *Check the 'Untranslated entries first' option from the View menu to get all untranslated strings at the top of the list.*

Translations with the **POEDIT** program are very simple and intuitive. When editing texts, user should pay particular attention for special characters that are described later in this document.

All translations of the source language that were modified during the program development can be marked as a '**fuzzy**'. In that case user can check the entry and decide if it's still correct or not. If the entry does not require correction it can be deselected by pressing the button.

In another case user should add correction, which will also mark the selected entry as translated.

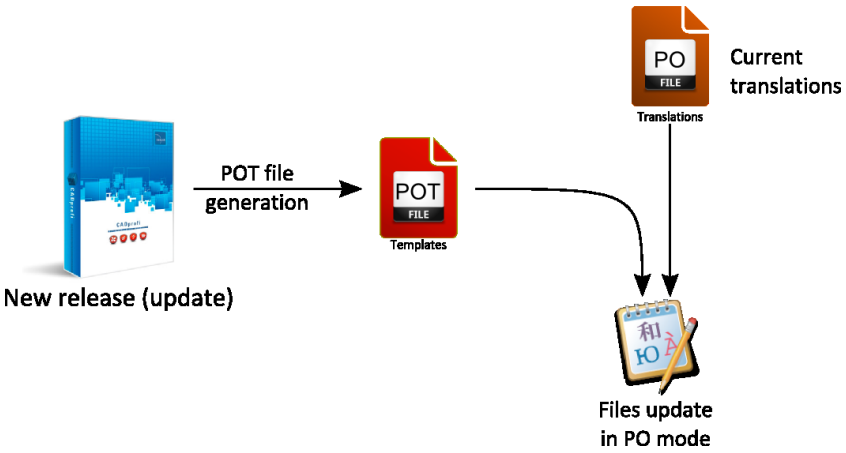
User can use the "**fuzzy**" mark while he translates the **PO** files on his own..

PO files update

It is possible that during translation, a new version of **CADprofi** program will appear. In such case, **PO** files will have to be updated. It can be solved in two ways.

The first option is to import to the updated program old ***.po** files (check the **Import PO files** page 102). In this way, the already made translations will be added to the program. Re-exporting **PO** files will include the already made translations as well as all texts which appeared in new program version.

Another way is to update the **PO** files by using templates (***.pot** files), that contain the current **CADprofi** dictionary. Thanks tot his it is possible to add to the already translated texts, all the texts that appeared in new program version.



PO files update process

Procedures

Update of PO files

1. Update **CADprofi** software to the latest version.
2. Generate **POT** files using the **Export** option to dictionary files.
3. Run the **POEDIT** and open the ***.po** file that you want to update.
4. From the **Project** menu select the **Update from .pot file** option.
5. Select the ***.pot** file and press **Open** to update.
6. All changes will be applied.

Import of PO files

Import of **PO** files is intended for program development and collaboration with translation agencies. For this reason, for standard user, the import **PO** files option is not available.
When importing **PO** files, all data is loaded into **internal CADprofi dictionaries**. User shouldn't change this data, because an eventual update may update and overwrite all changes, therefore the work may be lost.
It is completely different in context translation where all changes are saved in the '**UserData**' folder. A CADprofi update does not interfere with the data, so any user extensions or settings are automatically included in CADprofi program regardless of the installed version.

Procedures

Translation import

1. Run the CP_DICTIONARY command, after that an **Export/Import *.po files** dialog window will be opened.
2. Select the **Import *.po files to CADprofi dictionaries** option.
3. Check/uncheck dictionaries that you want to import.
4. In the **Select language** section, select the language that you would like to import.
5. Select the **Import/Do not import** option in the **What to do with 'fuzzy' texts** section (this option allows to specify if texts are marked as "fuzzy" should be imported to program dictionaries).
6. Click the **Import** button to close the window and start the procedure.

Files to translate:

cp_string.po – program interface
dictionary_CpTools.po –CADprofi tools
dictionary_Arch.po – CADprofi Architectural content
dictionary_Electro.po – CADprofi Electrical content
dictionary_Sanit.po – CADprofi HVAC & Piping content
dictionary_Mech.po – CADprofi Mechanical content
Other **PO** files – manufacturers' libraries dictionaries

Untranslatable words

Do not translate or change trademarks or program names such as:

- **CADprofi**, **AutoCAD**, **Bricscad**, **Cablofil** etc.
- phone numbers
- email addresses (e.g. **info@cadprofi.com** or **mailto:info@cadprofi.com**)
- folder names and ***.exe** files (e.g. **Common\cadprofi-connect.exe**).

These texts should be copied to translations or they should be left with translations.

Special characters

CADprofi program can be translated by editing **PO** files or by using the CADprofi Context translation method. In both cases, you should pay attention to special characters. Their presence is very important, and sometimes even critical User has to keep all special characters in his translation in the right order and position just like in the source text. User can only change the position of the percent '**%**' character (e.g. **%s**, **%d**, **%.10lf**, **%.12lg** etc.).

> ***Notice***
>
> ***Special characters must appear in the same form in both translations and the source language.***
> ***User can only change the position of the "%s" and "%d" characters to keep the logical meaning of the sentence.***

Translation example:

English	German
%s objects not found	*Keine %s Objekte gefunden*

Meanings of special characters that can be found in **CADprofi**:

\n – the end of line

**** – backslash

"" – means the ' "'

\[– means the '<'

\] – means the '>'

_ – means the space (for example at the end of the string)

%s, **%d**, **%10lf**, **%12ls**, **%10lg**, etc., – means the formatting string (%s = string, %d = value)

> ***Tip***
>
> ***Both POEDIT and CADprofi Context translation tool have internal procedures to keep special characters correctness, so pay attention for all messages displayed during the localization process.***

CADprofi® Architectural

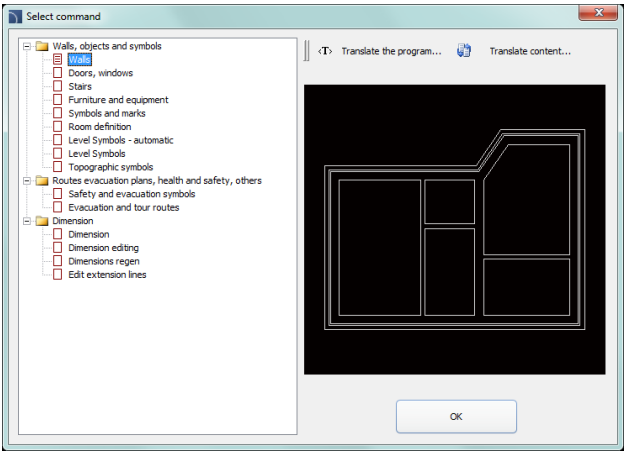
CADprofi Architectural

CADprofi Architectural helps user with creating construction plans, cross-sections and elevation views. It also allows to create composite walls, insert windows and doors, create architectural dimensioning, create descriptions and use objects from the furnishings and furniture library. A unique program feature is the possibility to design evacuation and safety plans. **CADprofi Architectural** is a sufficient addition to the base CAD programs, thanks to which user can create and edit building blueprints much faster.

CADprofi Architectural in 2 minutes

Architectural general commands

The **Architectural** command opens the dialog window from which we can choose the appropriate commands from the **Architectural** module. This command has been prepared for beginners. In the dialog window apart from the list of all commands we can also find a large preview window that shows the outline of the selected function.



Architecturalcommand dialog window

Procedures

Selecting the command

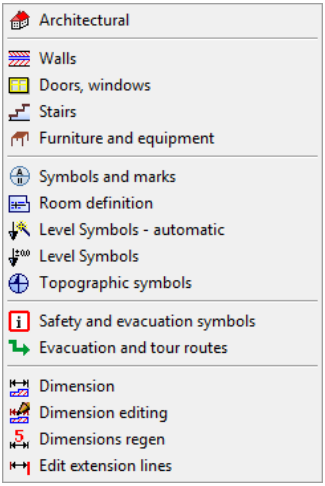
1. Run the **Architectural** command.
2. In the dialog window, expand the chosen category and select the needed command.
3. Selected command will start after clicking the **OK** button.

CADprofi Architectural – main program commands

CADprofi Architectural menu and toolbar

CADprofi Architectural commands

CADprofi Architectural toolbar



Menu of CADprofi Architectural module

Editing commands

Important editing commands used in designing can be found in the **CADprofi** menu and **CADprofi – Tools** toolbar.

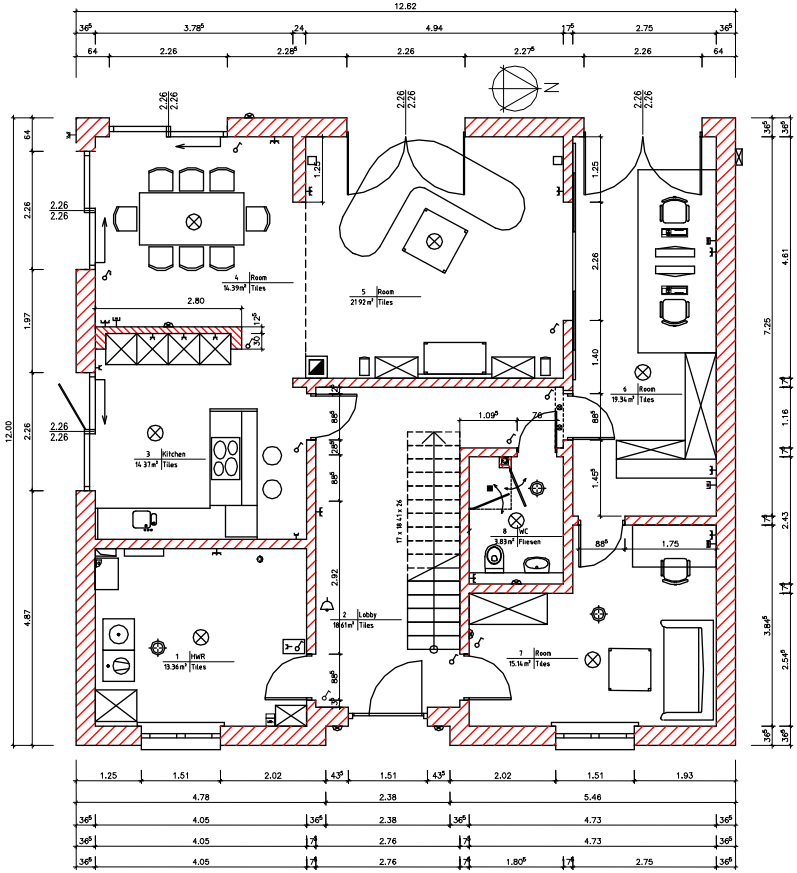
CADprofi – Tools toolbar

CADprofi Architectural – introduction

Creating architectural drawings is associated with drawing walls, inserting both woodwork objects and additional equipment as well as with describing drawings. **CADprofi** program offers the designer appropriate commands, which makes work easier and faster. Many options automate frequently performed actions, that's why it's very important to know how the program works.

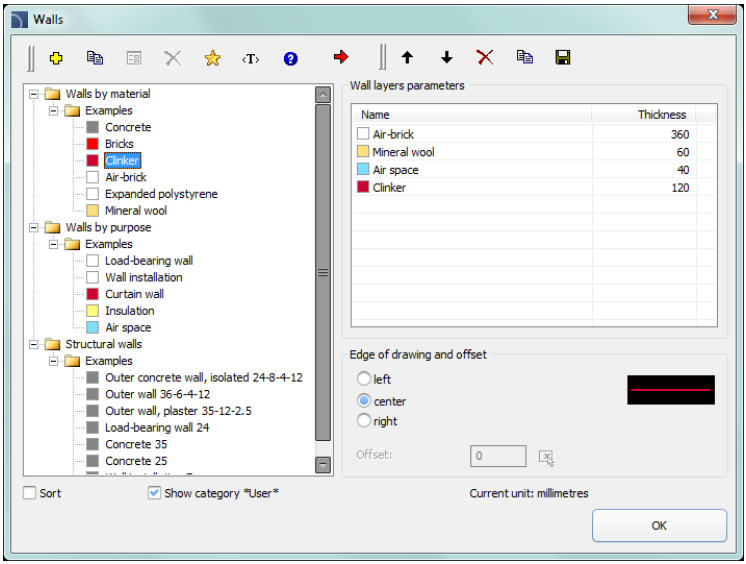
An important stage in the project creation is to describe the elements and rooms, so it is possible to create various BOMs.

After installing the program, the default unit is millimeters. Architectural designs are usually created in centimeters. User can change the design unit by running the **CADprofi - Options** command and in the **Units of drawing** field select **Centimeters**.



Drawing walls

In **CADprofi** program schematic walls are being drawn with lines that lie on appropriate layers. Composite walls can contain several layers that represent walls by material (brick, concrete etc.) or wall by purpose (load-bearing wall, insulation etc.).
Drawing single layer walls is possible after selecting the needed wall type and clicking the **OK** button. Such a process however does not allow to change drawing settings and wall thickness. These options are only available for walls that were added to the list that contain a wall definition. This list also allows for drawing of composite walls.
Due to very large variety of materials used in the construction branch, when starting your work with **CADprofi** it is recommended to define your own walls with your materials or structural elements.



Walls dialog window

The **Walls** dialog window contains the following elements:

Wall types menu:

- **Define new** – allows to add a new position to the list as a user wall
- **Copy** – copies the selected wall with all its parameters, therefore creating an element made by user. Copied lines may be edited.
- **Edit** – opens the edit properties window of a selected user created wall
- **Delete** – deletes the selected wall from the list.

- **Add to favourites** – adds the selected line to the ***Favourites*** (check page 122) list.
- **Add element to a list** – adds the selected line to the **List of wall layer parameters**.

Menu of wall layer parameters – allows to manage the content of the selected walls list.

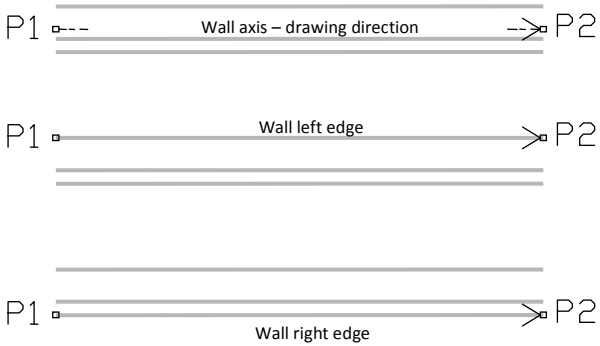
- **Move selected rows down** – moves the selected wall one position down.
  Move selected rows up – moves the selected wall one position up.
- **Remove selected rows** – deletes the selected wall from the list.
- **Copy selected rows** – copies the selected wall and adds it to the list.
- **Save as a structural wall** – allows to save the current list of selected walls as a structural wall.

Wall layer parameters – provides a summary of wall layers.

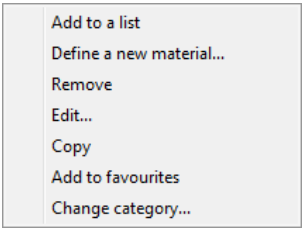
Thickness – specifies the thickness of specified layer in a composite wall.

Edge of drawing and offset – options that are used to determine the way of wall drawing and the offset.

- **Left/Center/Right** – drawing the wall with the axis or with an edge of the indicated points.
- **Offset** – an offset of the drawn lines from the points that are actually indicated in the drawing.



Pop-up menu – options menu for selected element/category which is available by clicking the right mouse button. Commands included in the menu are similar to the commands available on the dialog window toolbars.

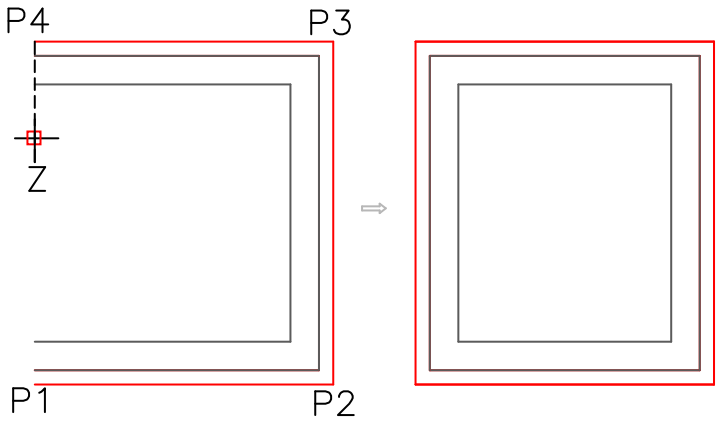


Procedures

Drawing the wall

1. Run the **Walls** command.
2. In the newly opened dialog window pull-down the category that contains the desired wall kind.
3. Select the desired wall type and click the **Add element to a list** button or double click on it in order to put it on the list.
4. (Optional) Repeat steps 2 and 3 in order to add more lines to the list.
5. (Optional) Using the **Copy selected rows**, **Remove selected rows** buttons add or remove the selected layer from the list.
6. Specify the **thickness** for each wall layer.
7. (Optional) Using the **Move selected rows down**/**Move selected rows up** buttons change the order of the lines on the list.
8. Specify the **edge of drawing and offset**.
9. Click the **OK** button in order to start drawing the wall.
10. Specify the starting point and next walls segment points (e.g. **P1**, **P2**, **P3**, **P4**). In order to finish drawing press the **Enter** or **Esc** key. It is also possible to close the command with the **Close** option (C key), thanks to this program will draw the last segment that will connect itself with the first segment of our drawn wall.

Drawing walls

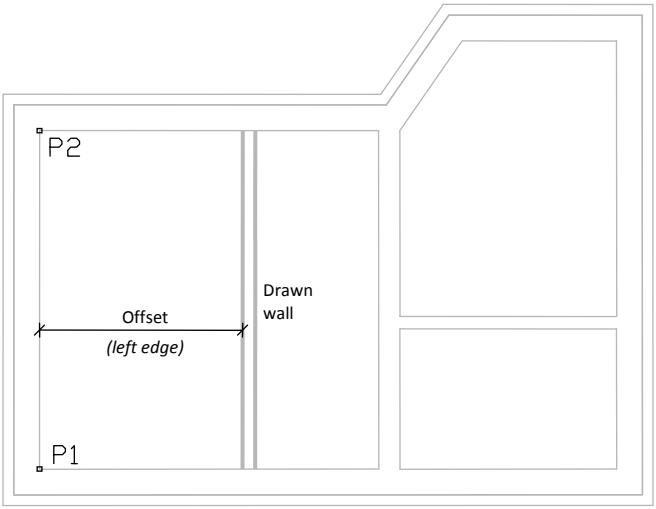


> ***Hint***
>
> ***Drawing with an offset can be used when drawing wall installations.***

Procedures

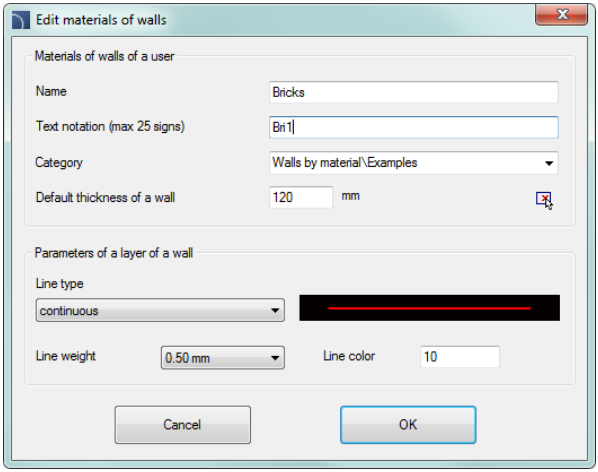
Drawing a wall installation with an offset

1. Run the **Walls** command.
2. Select the desired wall installation type and click the **Add element to a list**.
3. Specify the wall **thickness**.
4. Specify the **edge of drawing** as „left" and the **offset**, as e.g. 360 cm.
5. Click the **OK** button in order to start drawing the wall.
6. Specify the starting and the end points (**P1**, **P2**). The wall will be drawn with a specified distance from the indicated points.
7. Finish the drawing with the **Enter** or **Esc** key.



Edit walls materials

CADprofi allows users to define new walls types and to edit them.



Edit walls materials dialog window

The dialog window contains the following items:

Name – name of edited wall layer.

Text notation – text marking for lines used as an ending of layer's name in the extended layer structure (e.g. „concrete" wall type can lie on a **2DA_WAL_Bet** layer).

> ***Notice***
>
> ***It is recommended to use „unique values" of text notations in order to ensure proper recognition of lines by CADprofi.***

Category –category in which the selected wall type will be saved.

Default wall thickness – wall thickness that is used as a default value when the selected wall layer is being added to the **wall layer parameters** list.

Wall layer parameters – CAD settings for the layer, on which the selected wall type will be drawn. User should specify the color and both the line width and type for the appropriate layer.

Procedures

Creating new wall types in the database

1. Run the **Walls** command.
2. Click the **Define new** button in order to add a new wall type to the existing database. New element will be added to the ***User*** category. A new **Edit** window will open, in which user will be able to specify all wall parameters.
3. (Optional) Select an existing line and click the **Copy** button to create a new item. Selected line will be used as a template.
4. Click the **Edit** button in order to specify newly created wall parameters.

Edit wall type's data

1. Run the **Walls** command.
2. (Optional) If the wall type that you want to edit is located on the list of **wall layer parameters**, then you will need to remove it first from the list, to do this, select the element and click the button.
3. In the ***User***category select the wall type that you would like to edit.
4. Click the **Edit** button. A new edit dialog window will open.
5. (Optional) Specify the wall type **Name**.
6. (Optional) Specify the **Text notation** for the CAD wall layer (not recommended, if the element has been already used in the drawing).
7. (Optional) Specify the **Category**.
8. (Optional) Specify the default **thickness** for the selected wall type.
9. (Optional) Specify the **wall layer parameters**: type, color and line thickness.
10. Click the **OK** button to accept the changes.

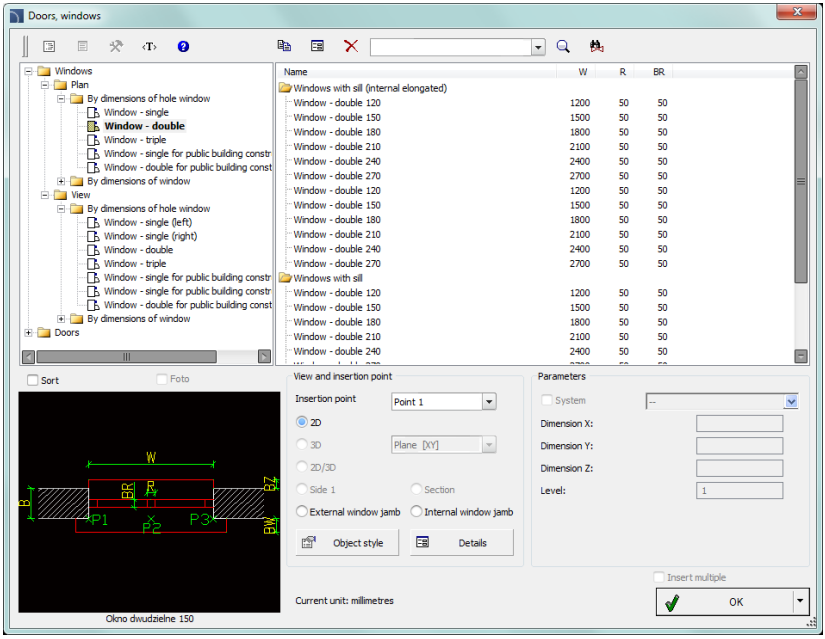
Notice

If user changes the parameters of CAD wall layer, which was previously added to the drawing, then in order to update it, user will have to draw a new wall segment with new settings.

Changing the text notation for walls that have already been inserted in the drawing will make them unknown for CADprofi.

Doors, windows

The **Doors, windows** command allows to insert doors and windows in both plan and view. For each objects databases are available that contain woodwork elements. All objects in the library are parametrical objects. It means that user has the possibility to specify all the dimensions when inserting objects to the drawing. This feature gives user a lot of opportunities because he is able to specify dimensions on his own, even for elements that are not in the standard program databases. If user inserts windows and doors in plan then he has to specify the wall thickness, in which he wants to insert the element. Providing the correct wall thickness will allow to automatically hide the wall lines, in which the element will be inserted.



Doors windows dialog window

The dialog window contains the following items:

List of categories – displays a list of available elements grouped in categories and subcategories.

Detailed list – contains a list of available element types with various dimensions.

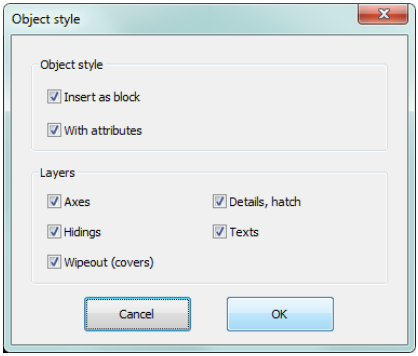
Preview – displays the preview of the currently selected element. Right-clicking on the preview window will zoom the view.

Sort – enables/disables categories alphabetical order.

View and insertion point – view selection and object insertion points. For all library objects a primary view is available (mostly 2D). Additional views are available only for some objects (e.g. **Internal window jamb** and **External window jamb** are available only for plan windows).

Insertion point – possibility to choose the objects base point (insertion point). Insertion points are marked on the preview as **P1**, **P2** etc.

Object style – options that allow to specify the object style and the level of object detail.



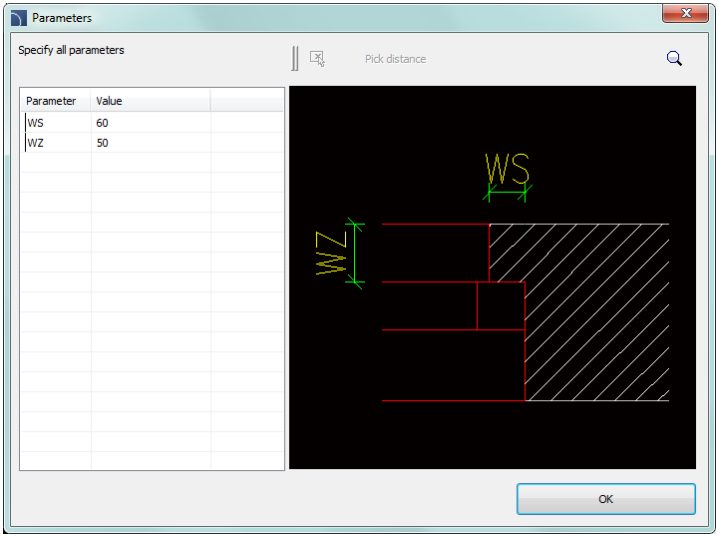
Object style dialog window

- **Insert as block** –allows user to insert an element that is exploded into component parts or a one single object (block).
- **With attributes** – enables/disables adding of attributes to the inserted object.
- **Axes** – enables/disables an option to draw axis in the inserted object.
- **Hidings** – enables/disables drawing elements that are hidden in an object.
- **Wipeout (covers)** – enables/disables addition to the inserted elements **Covers** (WIPEOUT) type objects.
- **Details, hatch** – enables/disables drawing of hatches and object details such as holes in the flanges, arrows that indicate the direction flow and others.
- **Texts**– enables/disables drawing of potential texts in objects.

Notice

All „Object style" settings will be remembered and used when user runs the command again.

Details – button that allows to specify dimensions for details that are visible in certain views such as **Internal window jamb** and **External window jamb**. Clicking the button will open a dialog window with visible current value dimensions.



Parameters dialog window

OK – closes the library and allows to insert the object into the drawing.

Insertion options – expands the list of available insertion options:

- **Insert (specify style for object)** – inserts selected object with a possibility to change options in the **Object style** dialog window.
- **Insert (specify all dimensions)** – inserts selected object with possibility to specify all dimensions that can be edited.
  After selecting this option a **Parameters** window will appear allowing user to specify values for each dimension. It is also possible to determine dimensions directly from the drawing. In order to do so, user has to activate the value edit field and then click the **Pick distance** button. It is also possible to save the user dimensions by Creating and extending the user type of series **Creating and extending user type of series** (check page 88).

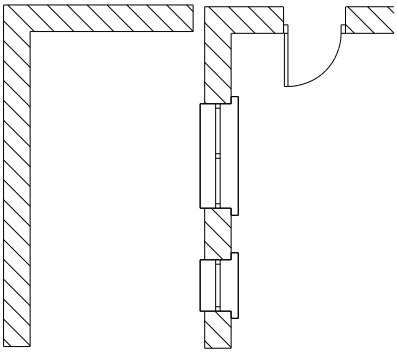
> ***Hint***
>
> ***By default, in the "Parameters" window displays the dimensions of the object selected in the command main dialog window.***

Procedures

Inserting doors and windows

1. Run the **Doors, windows** command.
2. In the dialog window choose the needed element (e.g. a window).
3. (Optional) Double click in the detailed list category in order to expand it.
4. Choose an element with appropriate dimensions from the detailed list.
5. (Optional) Select the **insertion point** for the selected view.
6. (Optional) Select the **view** for the selected object.
7. (Optional) Specify the settings in the **Object style** options.
8. Click the **OK** button in order to insert the element in the drawing.
9. (Optional) With the use of the button expand the list of insertion options and select **Insert (specify all dimensions)**, in order to specify the object dimensions.
10. (Optional) When inserting into a wall specify the object insertion point. If the wall is not detected, provide the wall thickness (**B** dimension) or indicate a point that specifies the thickness.
11. (Optional) Specify the rotation point of inserted object

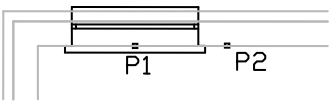
Doors, windows



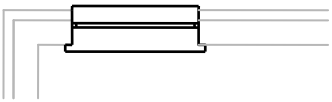
Walls view before and after windows and doors insertion

Inserting doors and windows without WIPEOUT

If the doors or windows are being inserted in the wall without hidings (without WIPEOUT), then after their insertion it may be needed to "cut" the wall lines with the help of the _TRIM command.



View after inserting a window

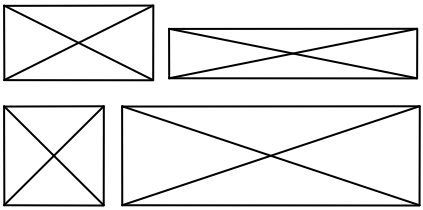


View after the wall is „cut"

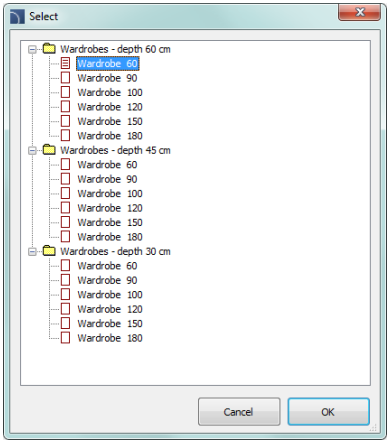
Furniture and equipment

The **Furniture and equipment** command possesses many objects that are used in construction projects. Furniture, sanitary equipment, home appliances and many more objects are available. This command also contains additional architectural elements such as: ventilation and exhaust, culverts and others.

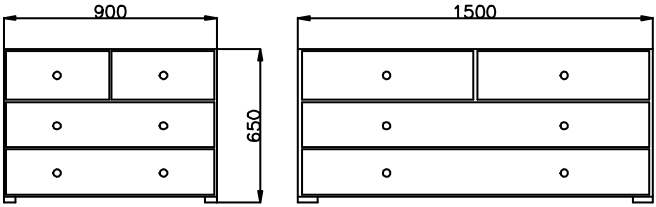
Objects available in this command are parametrical objects, for which user can specify all dimensions. Field used for specifying the dimensions are available in the dialog window. It is also possible to indicate dimensions from the drawing after clicking the **Pick distance** . For some objects, typical databases have been prepared that allow to quickly select typical elements after clicking the Types - database button.



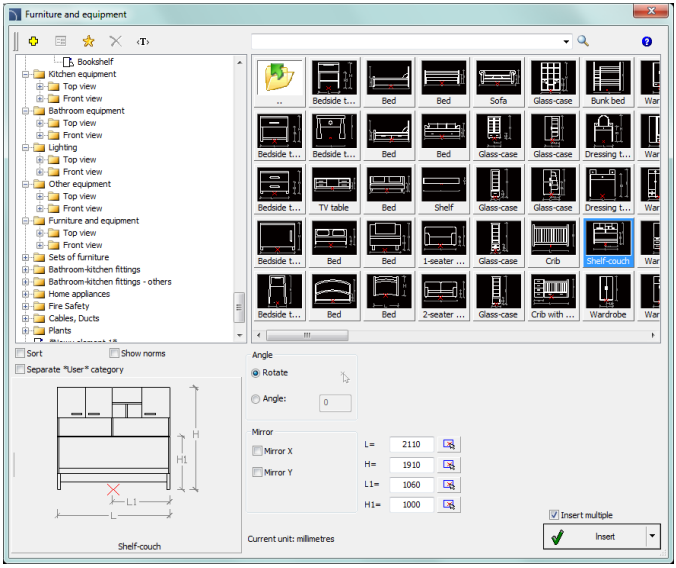
Different wardrobes dimensions from the databases of typical dimensions



Changing the dimensions for **CADprofi** objects is equivalent to rebuilding the block, not just with changing the scale in the **X** and **Y** directions. Thanks to this always the correct view is obtained and the specific elements (such as handles, holes etc.) are not deformed.



An example of a chest of drawers with a 900 and 1500 mm widths



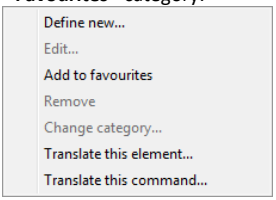
Furniture and equipment dialog window

The **Furniture and equipment** dialog window contains the following items:

Tools menu:

- **Define new** – allows to add a new element to the database. Read more about it in the **Defining user blocks** (check page 89).
- **Edit** – allows to **Editing user block parameters** (check page 92).
- **Add to favourites** – copies symbol to the ***Favourites*** (check page 122)
- **Delete** – deletes the selected symbols from the ***Favourites*** category.

Pop-up menu– options menu for the selected symbol, which is available after right clicking on the mouse button on the selected symbol. Commands in this menu are similar to the tools menu.



- **Categories tree** – a set of all categories and symbols in the library, shown as a pull-down tree. User can customize the display order by using the **Sort** option.

Thumbnail view – displays the content of currently selected category as thumbnails.

Preview – displays the preview of the currently selected element. Right-clicking on the preview window will zoom the view.

Angle/Rotation – possibility to specify a fixed angle when inserting an element or enabling the rotation option, in which the element's angle is defined by rotation during insertion

Mirror X, Y– enabling this option will create an X or Y mirror image for the specified object.

Insert multiple– enables/disables the possibility of inserting multiple symbols to the drawing. The insertion process requires from the user to press the **Enter** or **Esc** key when he finished inserting multiple symbols

Procedures

Inserting furniture and equipment

1. Run the **Furniture and equipment** command.
2. In the dialog window choose the needed category.
3. Select the object that you would like to insert. In the bottom-left corner of the dialog window a preview of the selected object will be displayed.
4. (Optional) Specify object dimensions. Dimensions can be picked up from the drawing by pressing the button.
5. Specify the insertion parameters: **System**, **Rotation**, **Scale**.
6. Click the **Insert** button.
7. Specify the insertion point.
8. (Optional) Specify the rotation angle of the inserted object.

Furniture and equipment

Management of elements content

CADprofi program possesses several thousands of elements grouped in appropriate categories. Witch such a large number it is sometimes hard to find elements needed at the moment. On the other hand, when working on a particular project it is usually not needed to work with more than several dozens of elements. In order to speed the work, user can add selected items to the ***Favourites*** category thanks to which he will be able to get a much faster access to them.

It is possible to create sub-categories related to a specific theme, such as a separate category for elements used in used in apartments construction and others in designing offices or social rooms. Furthermore, one element can be added to many sub-categories.

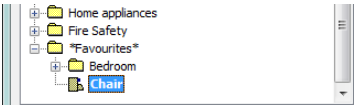
For elements placed in the ***Favourites*** category it is possible to change their name, which will be later used in specifications, legends and descriptions. Change of name can be also used to user own translations for some symbols from international standards.

Adding symbols to favourites

Add to favourites button allows to copy any symbol to the ***Favourites*** category. This folder is created automatically after user has copied the first symbol.

Changing the element name

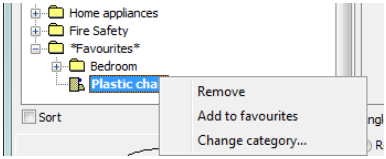
Initially, each copied symbol receives the default name from the original item.
In order to change the element's name user should first select it and then press the **F2** button to activate the edition mode.



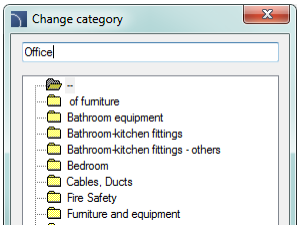
> ***Notice***
>
> ***Changed name will be visible in tables and descriptions that are created with the „Bill of materials" (check page 46) command.***

Changing category

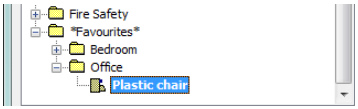
In order to change or define a new category, click the right mouse button on a desired symbol and from the context menu choose the **Change category** option.



In the dialog box, user can select an existing category or enter a new category name. After accepting, selected element will be moved to the defined category.

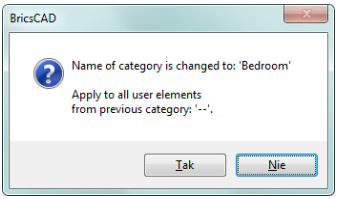


CADprofi application allows to create many copies of each element and it is possible to place them in different categories. Thanks to this user can create categories that group elements of a specified type or the most common used elements for a given industry type.



Changing the category or many symbols

If user is editing a symbol in his native category in which there are many elements, then **CADprofi** application will change category for all these elements.

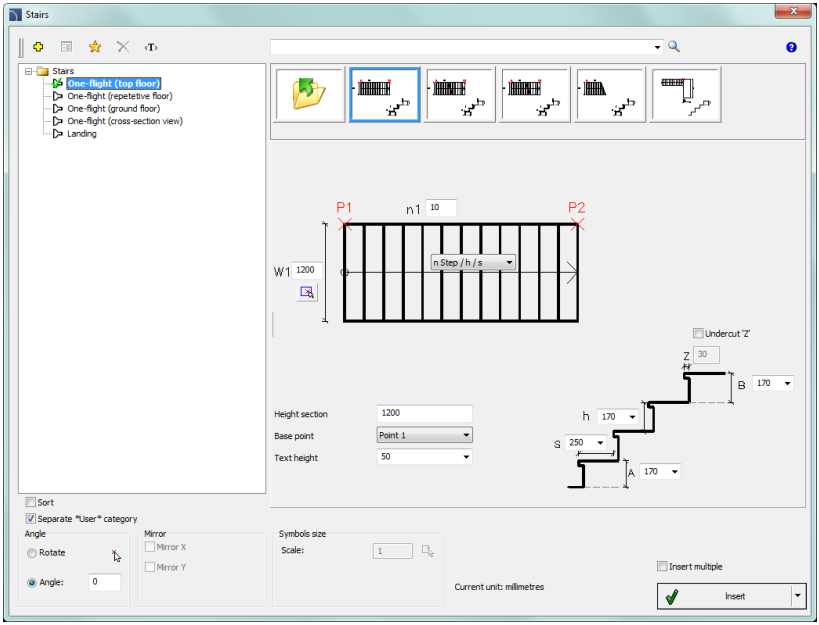


Deleting elements

It is possible to delete elements from the ***Favourites*** category. To do so, select the element and choose the **Delete** option from the dialog window.

Stairs

The **Stairs** command allows to quickly create stairs with specified parameters. Both one-flight and landing are available. From these elements it is possible to create stairs with any number of cases.



Stairs dialog window

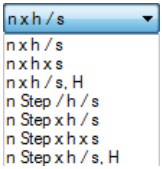
Basic functionality and options regarding work with objects are described in the **Furniture and equipment** (check page 120) chapter. Additional options that are included in the command:

Preview and parameters – an example preview with element parameters.

- **n1** – number of treads.
- **W1** – total stairs width.
- **S** – tread depth.
- **H** – tread height.
- **A** – first thread height.
- **B** – last thread height.
- **Z** – „undercut" tread value.
- **Undercut „Z"** – enables/disables drawing of „undercut" for treads.
- **Height section** – the actual value of the stairs cross-section view. The staircase view (both normal and treads hiding) depends on this height.

- **Base point** – the choice of object insertion points (**P1**, **P2** on the preview).
- **Text height** – text height that is used for stairs description.
- **Notations** – specifies the stairs notation type.
  The list of available stairs notations.

  Selecting an empty field located on the end of the list, means that no notation will be inserted.

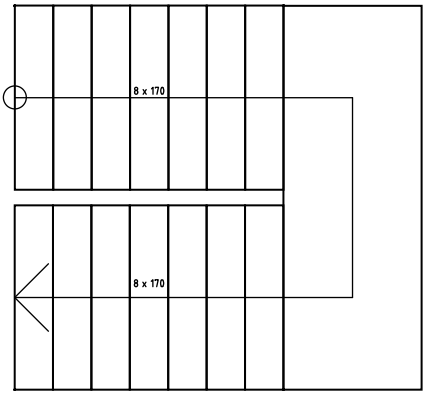


Procedures

Inserting stairs

1. Run the **Stairs** command.
2. In the **Stairs** dialog window on the **List of elements** click on a stair type that you would like to insert into the drawing.
3. (Optional) Select the stair type from the list of available stair's preview located on the top of the window.
4. In the preview window specify parameters for stairs that are going to be inserted.
5. (Optional) Change or indicate the rotation **angle**.
6. (Optional) Enable the **Rotation** function in order to specify the rotation angle when inserting the block.
7. (Optional) Enable/disable the **Insert multiple** option
8. Click the **Insert** button in order to insert the element into the drawing.
9. Indicate the insertion point for block.
10. (Optional) Indicate two points that specify the block rotation angle.

Inserting stairs



Double staircase created from available elements

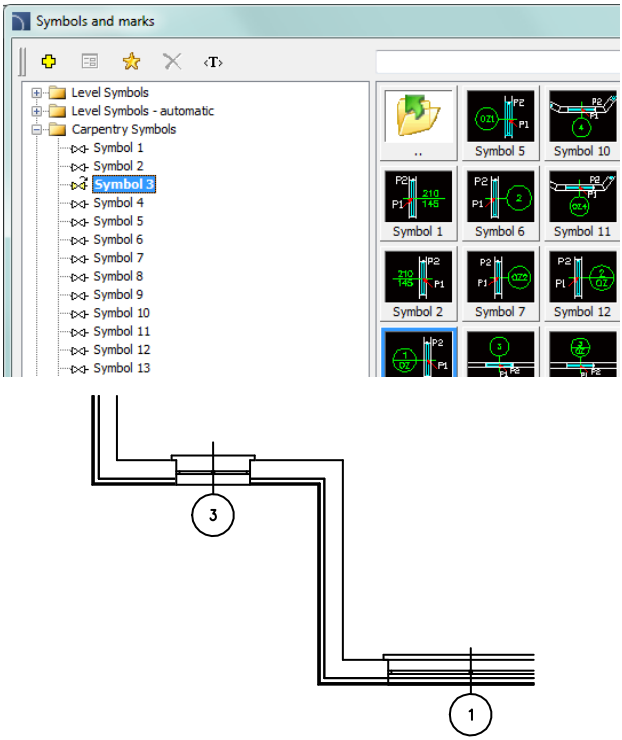
Symbols and marks

CADprofi possesses several categories architectural marks symbols. Most of these symbols are available in the **Symbols and marks** command. To obtain a quicker access for certain marks, they have been also added to additional commands e.g. **Level symbols** or **Topographic symbols**.

One of the categories available in the **Symbols and marks** command are the woodwork elements marks. These symbols allow to specify number, label and dimensions of described element. These data can be later used when creating BOMs and obtaining a variety of information about the project. In order to describe various objects user should use appropriate symbols.

Example: for windows inserted into the drawing horizontally, user should use appropriate symbols and for windows inserted with a 90 degrees angle - different symbols.

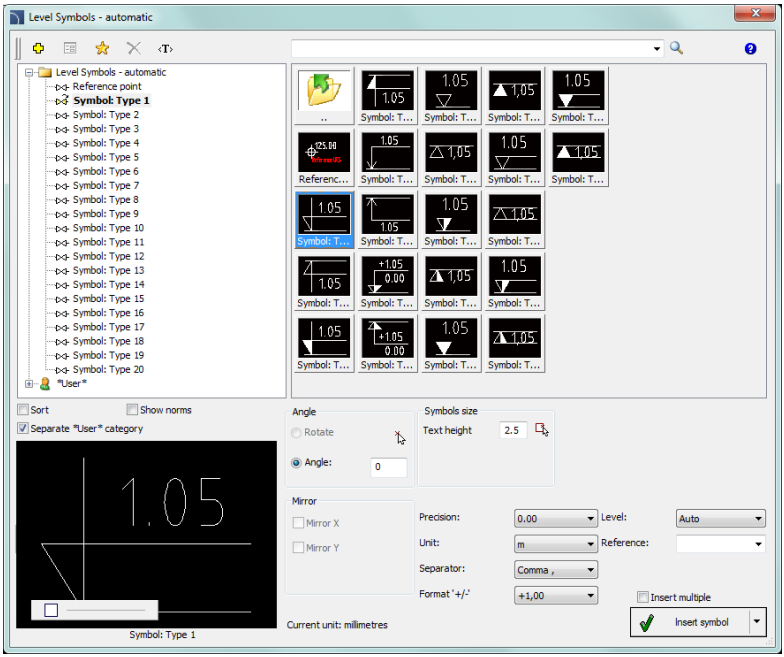
When describing many objects it's convenient to select the **Insert multiple** option.



Symbols and marks

Level Symbols - automatic

The **Level Symbols - automatic** command allows to insert height points with automatic filling of elevation values for indicated points.



Level Symbols – automatic dialog window

Options available in this dialog window have been described in previous chapters (check page 120). Additional options included in this command:

Text height – specifies the marks text height and scale. User can enter a value or indicate it from the drawing (indicate two points).

Precision of dimensions – defines the decimal number of the ordination values in the symbol that's going to be inserted.

Unit – determines the unit, which will be changed into the elevation value in the inserted symbol. **CADprofi** program determines the actual ordination value in accordance with the currently drawing units that were set in the **CADprofi - Options** command. The **Unit** option allows to recount the actual height into one of available units.

Separator – determines the decimal number of ordination value.

Format '+/–' – determines if the positive ordination values should be preceded with a „**+**".

Level – determines the reference point of the symbol ordination value. The following ways to determine the coordinate system are possible:

- **Auto** – calculates the ordination value in regards with the previously set **Reference point**.
- **Auto WCS** – calculates the ordination value from the 0.0 point of the World Coordination System.
- **Auto UCS** – calculates the ordination value from the 0.0 point of the User Coordination System.

Reference point – additional text that informs for example about the above sea level height or the ground foundation height.

Procedures

Inserting the reference point

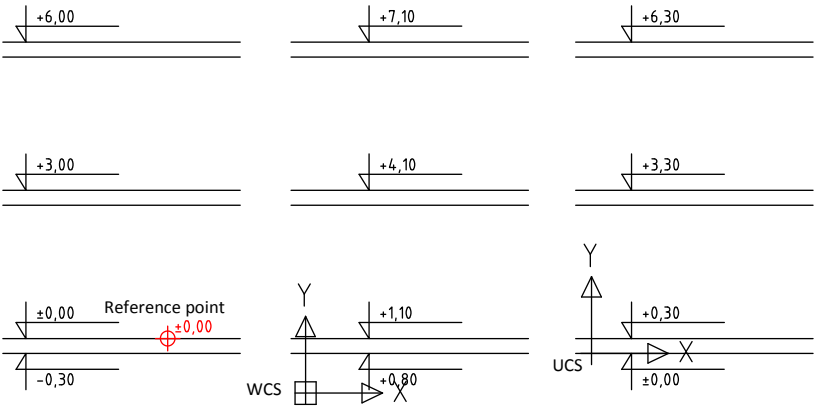
1. Run the **Level Symbols - automatic** command.
2. In the dialog window select the **Reference point**.
3. (Optional) Specify the symbol text height.
4. (Optional) Specify the following options: **Precision**, **Unit**, **Separator** and **Format '+/–'**.
5. Choose the coordination system, which will specify the point **Level** or enter the value manually. Reference point is a benchmark for other height points that's why this point **Level** is specified as „**0**".
6. Click the **Insert symbol** button to insert a symbol into the drawing.

> ***Notice***
>
> ***Only one reference point can be added to the drawing, making an attempt to insert the next point will display an appropriate message and will allow to replace the old reference point.***

Inserting an automatic level symbol

1. Run the **Level Symbols - automatic** command.
2. In the dialog window select the appropriate level symbol marking.
3. (Optional) Specify the symbol text height.
4. (Optional) Specify the following options: **Precision**, **Unit**, **Separator** and **Format '+/–'**.
5. Specify the **Level**, from which the current level will be count. If the level should be counted from the reference point select the **Auto** option.
6. (Optional) Define or select the **Reference point** for the selected point.
7. Click the **Insert symbol** button to insert a symbol into the drawing.

Symbols and marks



Ways to specify the level: Reference point, WCS and UCS

Level Symbols

The **Level Symbols** command allows to insert height points. The only difference regarding the **Level Symbols-automatic** lies in the fact than in the **Level Symbols** user can manually specify the level symbol and any additional values.

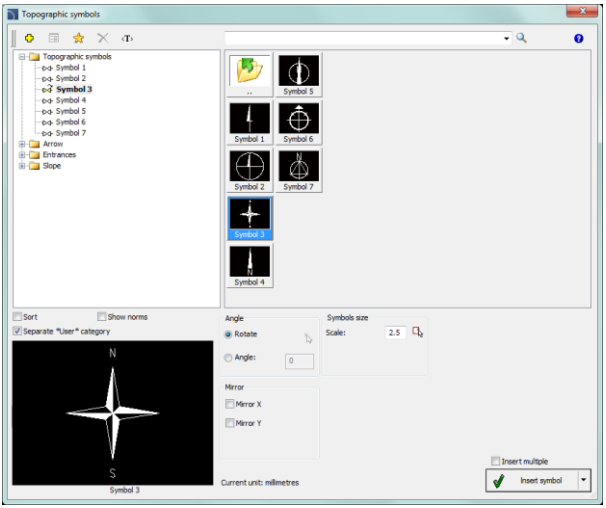
Procedures

Inserting a level symbol

1. Run the **Level Symbols** command.
2. In the dialog window select the appropriate level symbol marking.
3. (Optional) Specify the symbol rotation angle.
4. (Optional) Specify the symbol text height.
5. Specify the **Level** value, optionally choose additional +/– markings.
6. (Optional) Specify or choose the **Reference point** for the chosen point.
7. (Optional) In the symbol preview enable/disable an option that allows to draw a bottom line in description.
8. Click the **Insert symbol** button to insert a symbol into the drawing.

Topographic symbols

The **Topographic symbols** command allows to insert into the drawing many different symbols such as: arrows, cardinal direction symbols, slopes and others.



Topographic symbols dialog window

Options available in this dialog window have been described in previous chapters (check page 120).

Procedures

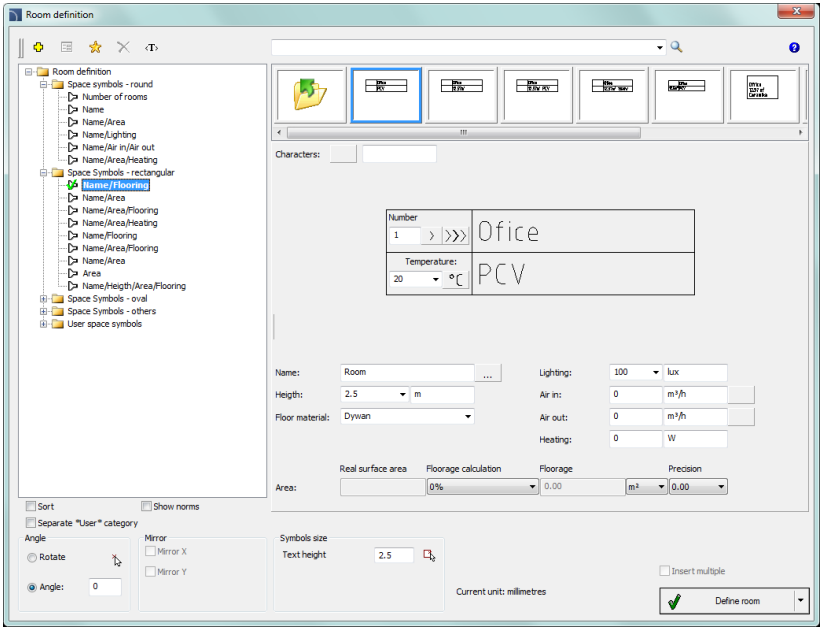
Inserting topographic symbols

1. Run the **Topographic symbols** command.
2. In the dialog window select the appropriate symbol.
3. (Optional) Specify the symbol **rotation angle**. You can enter the value or press the button in order to indicate two points from the drawing that define the rotation angle.
4. (Optional) Select the **Rotation** option in order to specify the rotation angle when inserting a block into the drawing.
5. (Optional) Specify the symbol size by entering the **scale** value or press the button in order to get the scale from another symbol that's located in the drawing.
6. (Optional) For certain symbols you can specify additional parameters, e.g. for slope symbols you can specify the slope value in percent or in degrees.
7. Click the **Insert symbol** button to insert a symbol into the drawing.

Symbols and marks

Room definition

In **CADprofi** it is possible to define and describe rooms or other usable areas. Program can automatically find boundaries and calculate their surface. It is also possible to manually define the rooms or to take over "foreign" boundaries (closed polylines), which is useful when working on architectural drawings that are made outside of **CADprofi**.
Program also allows to create room specifications, that's why when user defines a room, he should also specify their name, number and additional data.



Room definition dialog window

Options available in this dialog window have been described in previous chapters (check page 120). Additional options included in this command:

Number – defined room number. This field is automatically filled with a first free number. Thanks to available buttons, user can choose between first available free number ❯ and a next number ❯❯❯. This field can be filled with any value.

Information about defined room – a collection of information about defined area. These data will be saved as a collection of attributes of selected room label. The **Browse** ... button allows to select typical data for rooms that have specific purpose.

Real surface area – field that shows information about actual room surface area.

Calculating the floorage area:
Program automatically calculates floor area of defined room. Typically, construction projects do not include the plasterwork thickness. Therefore, when calculating floor area program can take into account "plasterwork" deductions.
It is also possible to manually set the floor area (e.g. if it's needed to deduct the area under the low-sloped roofs in attics). When calculating the surface it is very important to have the same design unit as the one that we used in the drawing. User can specify the area format with the use of **Precision** and **Floor unit**.

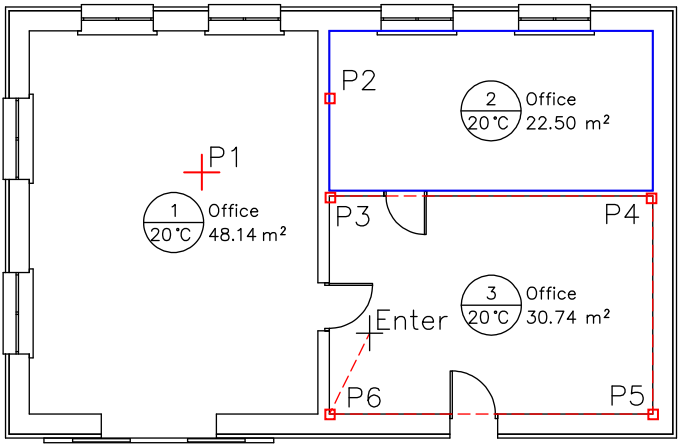
Ways to define rooms:
After specifying the parameters user should define them on the drawing. Button that is used for it has got a drop-down list that allows to select a method of determining the boundaries:

- **Define room (automatically)** – this option will automatically search for a closed area, in which the room has been defined. This method can be used when all walls have been drawn by **CADprofi** program and they haven't been interrupted for example when inserting doors or windows (in **CADprofi** program there is no need to interrupt the walls, if windows and doors are being inserted with hiding objects - WIPEOUT).
  Automatic search for room boundaries can also be used even if the walls weren't drawn by **CADprofi** program on condition that they are placed on the „2DA_WAL_" layer.
- **Select boundary** – this option allows to indicate a boundary (closed polyline), which will be acquired by **CADprofi** program as a room. This option is often used when working on "foreign" construction drawings, in which the boundaries that restrict the room are already drawn.
  Notice: in CAD programs polyline is closed if user completed its drawing by using the **Close** command. This status can be checked in the CAD properties window.
- **Outline a room** – possibility to drawn a boundary through manual outline. User should indicate all room corners. In order to finish this command user should press **Enter** or right mouse button.
- **Dividing line of rooms** – draws the room dividing line. Diving lines are being inserted on a layer which is disabled by default in printing. This line allows to add a logical division or a room without need to draw any additional walls.

Procedures

Room definition

1. Run the **Room definition** command.
2. In the dialog window choose the needed room label.
3. Specify the number, name and additional room data.
4. Specify the **Floorage area** options.
5. Press the **Define room**, in order to indicate a point in the drawing (**P1**) for automatic determination of room boundaries.
6. (Optional) Expand the button and select the **Select boundary** option in order to indicate a boundary on the drawing (**P2**), which determines the room.
7. (Optional) Expand the button and select the **Outline a room** option in order to indicate all room corners (**P3**, **P4**, **P5**, **P6**). **Enter** key ends the command.



Notice

For automatic boundary search, the indicated point must be located inside a "closed" area.
The room can have only one label.

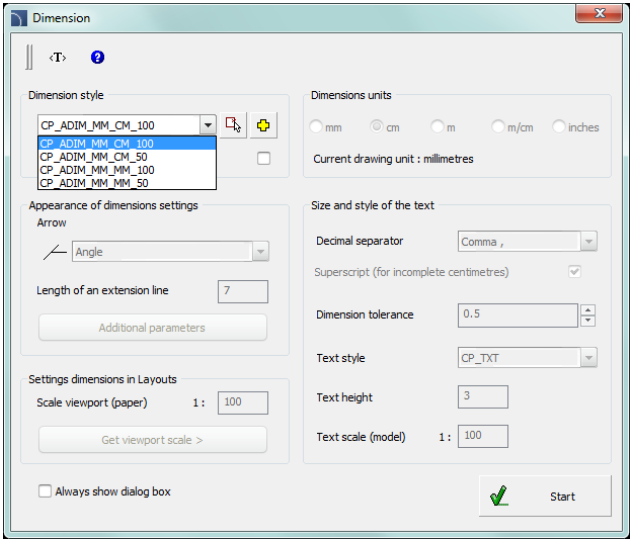
After making changes in the drawing, program will automatically update the floorage area calculations (if user doesn't want to update the floorage area, then he should determine the floorage area manually).
During the update, room boundaries, with removed labels will be deleted.
Data is updated each time the user presses the **Define room** command.

Architectural dimensioning

Dimensions

The **Dimension** command is used for easy dimensioning with styles that suit the construction projects. It is possible to dimension in model or paper area. This command allows to freely customize the existing styles, or define a user own dimensioning style. What is more, drawing can be prepared in a different unit (e.g. mm) and dimensions can be created in a different unit (e.g. cm).

Program also allows to change the "normal dimensioning" into "architectural dimensioning".



Dimension dialog window

The **Dimension** dialog window contains the following elements:

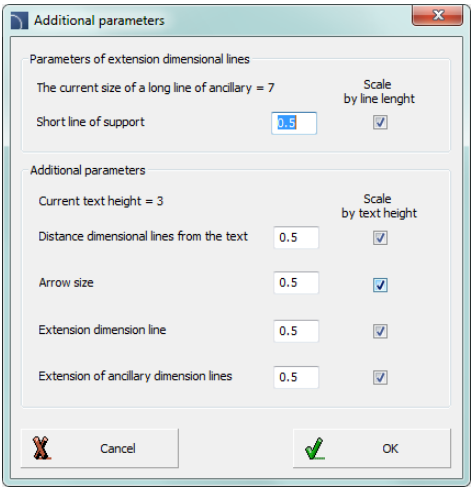
Dimension style – list of **CADprofi** styles and styles created by the user.

- **Get style from drawing** – allows to choose a dimension style by selecting an object from the drawing.
- **Create a new dimensioning style** – creates a new dimension style.
- **Change parameters of selected style** – enables the possibility to edit the selected style settings. Changes will only be made in the selected style.

Appearance of dimensions settings – a collection of settings that define the selected style:

- **Arrow** – allows to select the type of the main dimension line endings of the selected style.

- **Length of an extension line** – extension line length measured in drawing units. This value is scaled accordingly to the set text scale.
- **Additional parameters** – opens the dialog window that contains additional settings related with dimensions appearance.



Additional parameters

Settings dimensions in layouts – during dimensioning in paper area (sheets) program will calculate the dimensions accordingly to the viewport scale. Therefore it is needed to enter the **viewport (paper) scale** or to get it from the indicated viewport in the sheet.

Dimensions unit – program calculates dimensions from the current unit into the selected unit (below the current drawing unit is being displayed). Basic units such as mm, cm, m and inches are available. Additional option "m/cm" work in a specific way - if the dimension is less than one meter then the dimension text only includes centimeters (e.g. for 87,5 cm the dimension will include the 875 text), if the dimension has got more than one meter, then the dimension text will include both meters and centimeters (e.g. for 187,5 cm the dimension will include the 1.87^{5} m text).

Size and style of the text – settings used for the dimensions text display.

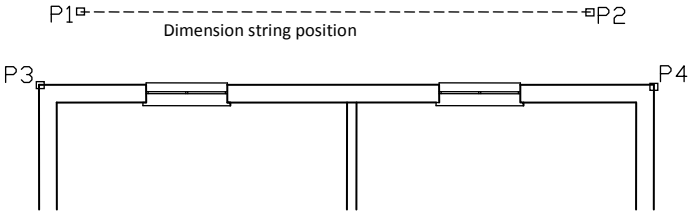
- **Decimal separator** – symbol oddzielający miejsca dziesiętne.
- **Superscript (for incomplete centimeters)** – enables/disables the display of decimal values as a superscript.
- **Dimension tolerance** – a value to which all dimensions will be rounded up.
- **Text style** – gives the possibility to choose one of available text styles.

- **Text height** – text height in mm in printings.
- **Text scale (model)** – printing scale for selected drawing. This option is used only for appropriate text adjustment.

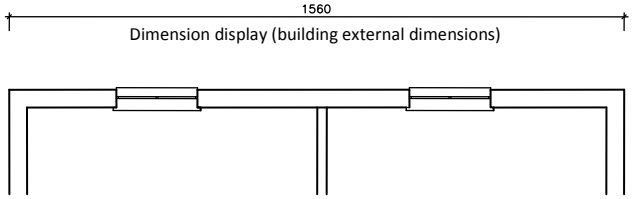
Always show dialog – enables/disables the display of the dialog window after pressing the **Dimension** command. If the window is not being displayed and it is necessary to change the settings then user can enable the dialog window with the **Settings** option (S key).

Dimensioning is carried out in two stages:

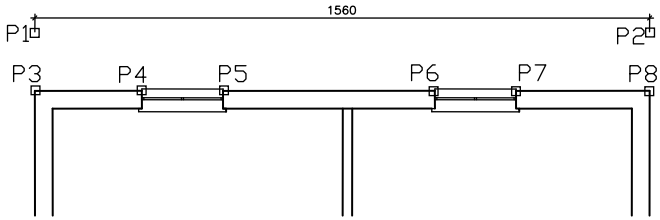
- **In the first stage** user should specify two points that will determine the dimension string position (**P1**, **P2**).



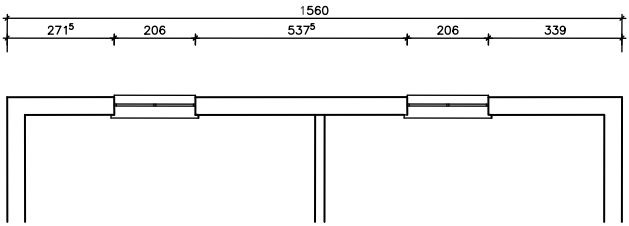
- **In the second stage** user should indicate additional measured points (**P3**, **P4** etc.). In one string user can specify any number of dimensions. In order to complete the dimensioning process press the **Esc** key or click the right mouse button.



After running the command again user can add additional dimensions to the drawing. When determining the dimension string user should indicate the side points (**P1**, **P2**) of the previous dimensioning. In this way user can attach additional strings. After indicating the string location, user should indicate additional measured points (**P3**–**P8**).



After running the command twice the following result appears:



Procedures

Architectural dimensioning

1. Run the **Dimension** command.
2. (Optional) If the dialog window didn't appear, select in the command line the **Settings** option (S key).
3. In the dialog window select the dimensioning style.
4. (Optional) Click the **Create a new dimensioning style**, to add a new style to the current drawing.
5. (Optional) Enable the option **Change parameters of selected style**, in order to make changes in the dimension styles settings.
6. Click the **OK** button to close the dialog window and start dimensioning.
7. (Optional) In the command line select the **Object** option (O key) and indicate an existing dimension from the drawing. It will allow to continue an existing dimension string.
8. Indicate the first point, where the dimension string will be located.
9. Indicate the second point, where the dimension string will be located.
10. Indicate the first and second measured point. The first dimension will be created.
11. (Optional) Indicate additional points in order to add additional dimensions to the string.
12. Press the **Esc** key or click the right mouse button to end the command.

Architectural dimensioning

Notice

After editing the drawing, for example when user stretches the dimensions it is needed to regen the dimension texts. Regeneration is performed by running any CADprofi dimensioning command.

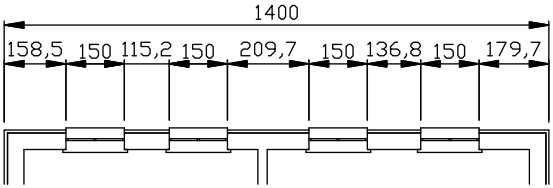
Dimensions editing

The **Dimensions editing** command is used for changing the style of existing dimensions that have been already added to the drawing. User can select or create a new style and then and afterwards indicate dimensions on the drawing for which the selected style will be applied. In this way it is possible to replace "normal" dimensioning into architectural dimensioning.

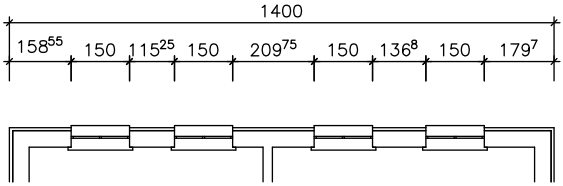
Procedures

Dimensions editing

1. Run the **Dimensions editing** command.
2. In the **Dimensions editing** dialog window select the dimension style that you would like to apply to existing dimensions.
3. (Optional) Click the **Create a new dimensioning style**, to add a new style to the current drawing.
4. (Optional) Enable the option **Change parameters of selected style**, in order to make changes in the dimension styles settings.
5. Click the **Replace** button to close the dialog window and enable the dimensions selection.
6. Select dimensions, which style you would like to replace.
7. Click the **Enter** key or right mouse button to make the changes and end the command.



Normal dimensioning



View after changing the normal dimensioning into architectural dimensioning

Dimensions regen

The **Dimension** command allows to change dimensions styles. These changes will apply to new dimensions, but won't affect dimensions that already exist in the drawing. In order to apply changes to all dimensions it is required to run the **Dimensions regen** command. .
This command also updates dimension texts after editing the dimensions geometry (e.g. during dimension stretch).
This command does not display any dialog window, therefore any results of this command can be read only in the CAD program command line.

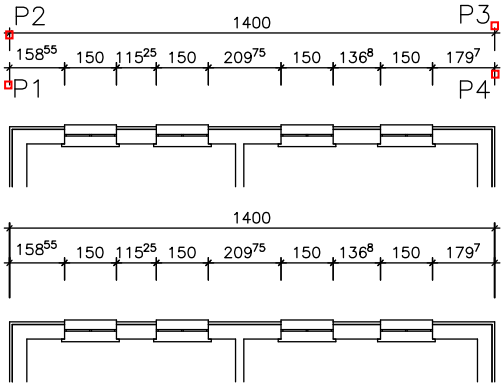
Edit extension lines

In architectural dimension strings the side dimension lines are usually used to "link" them with additional strings. Therefore, the side extension lines should be longer than other lines. The **Edit extension lines** command is used to shorten or lengthen the dimension side extension lines. The need to conduct such an operation may be needed when user makes various editing changes that regard dimensions such as: deleting, moving, dimensions copying etc.

Procedures

Edit the length of extension side lines

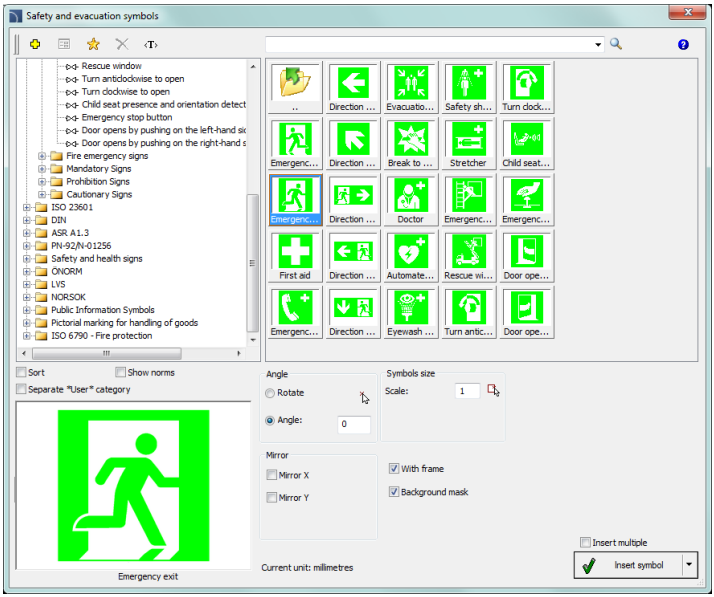
1. Run the **Edit extension lines** command.
2. Click on any dimension that was created with **CADprofi** in order to shorten/lengthen the side extension line.
3. (Optional) Click on next dimension to edit it.
4. Press the **Esc** key or click the right mouse button to end the command.



View before and after the side extension lines were lengthen

Safety and evacuation symbols

The **Safety and evacuation symbols** command allows to insert to the drawing a variety of markings used in evacuation, fire safety and other plans. This command also contains public information symbols as well as many more.



Safety and evacuation symbols: dialog window

Options available in this dialog window have been described in previous chapters (check page 120). Additional options included in this command:

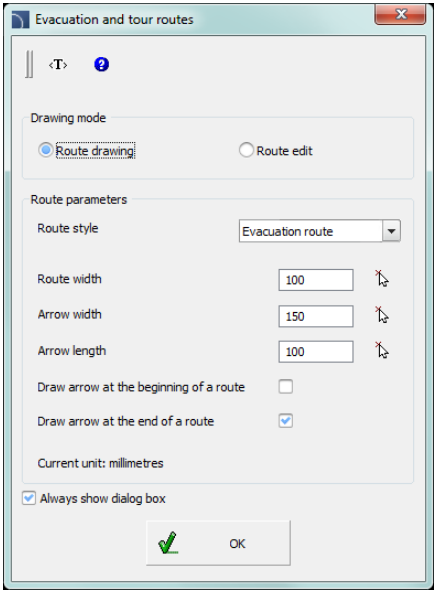
With frame – enables/disables drawing a frame around the inserted symbol. The frame can be useful, for example, in case of necessity of hatching the area where the symbol is located. Thanks to the frame the symbol won't be blended with the hatched area.

> ***Notice***
>
> ***Many "evacuation" symbols contain small details. In order to enhance the details of these symbols a special outline of them is placed on the CPCOLOR_255 layer. During printing user must adjust both the width and colour of this layer. Alternatively, user can disable this layer, so it won't be printed.***

Safety and evacuation symbols

Evacuation routes

The **Evacuation and tour routes** command allows to draw and edit evacuation and fire safety routes as well as other routes such as: museum and gallery sightseeing tours etc. At the end of the route user can insert arrows that indicate directions.



Evacuation and tour routes dialog window

The dialog window contains the following items:

Drawing mode – selection of route **drawing** or **editing** mode.

Route style – selection of route type. Each route type is created on a separate layer.

Route options – gives the possibility to specify route width and the dimensions of arrows.

Draw arrow at the beginning/end of a route – enables/disables drawing of arrows at the route beginning/ending.

Always show dialog window – enables/disables the display of dialog window after running the **Evacuation and tour routes** command. If the window is not displayed then the route will be drawn accordingly to the previously selected settings. In order to enable the display of dialog window user should select the Settings option (S key).

Procedures

Drawing routes

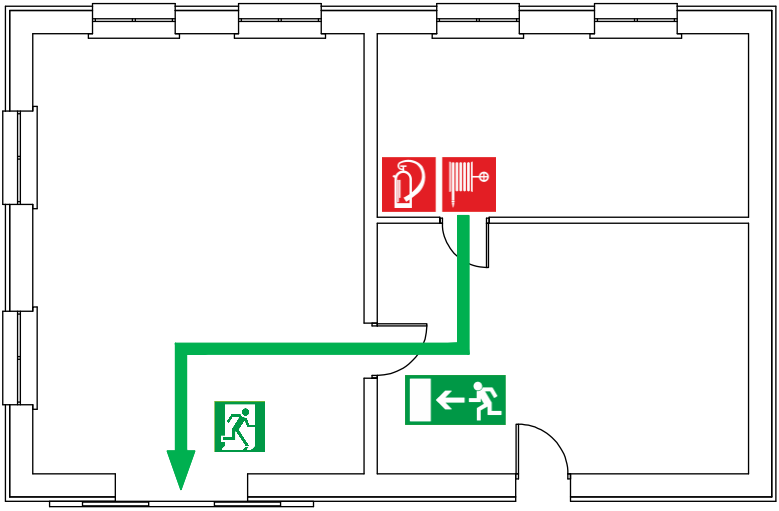
1. Run the **Evacuation and tour routes** command.
2. In the dialog window select the **Route drawing** mode.

3. Select the **Route style**, e.g. „Evacuation route”.
4. (Optional) Specify the **arrow width/height** settings.
5. (Optional) Enable/disable the **draw arrow at the beginning/end of a route** option.
6. Press the **OK** button to close the dialog window and start drawing the route.
7. Specify next points for the route. You can finish the drawing by pressing the **Enter** key or clicking the right mouse button.

Editing routes

1. Run the **Evacuation and tour routes** command.
2. In the dialog window select the **Route edit** mode.
3. Select the **Route style**, e.g. „Evacuation route”.
4. (Optional) Specify the **arrow width/height** settings.
5. (Optional) Enable/disable the **draw arrow at the beginning/end of a route** option.
6. After pressing the **OK** button select the route to which you would like to apply new settings. The indicated route will be changed.

Evacuation routes



An example of an evacuation route and evacuation and fire safety symbols.

CADprofi® HVAC & Piping

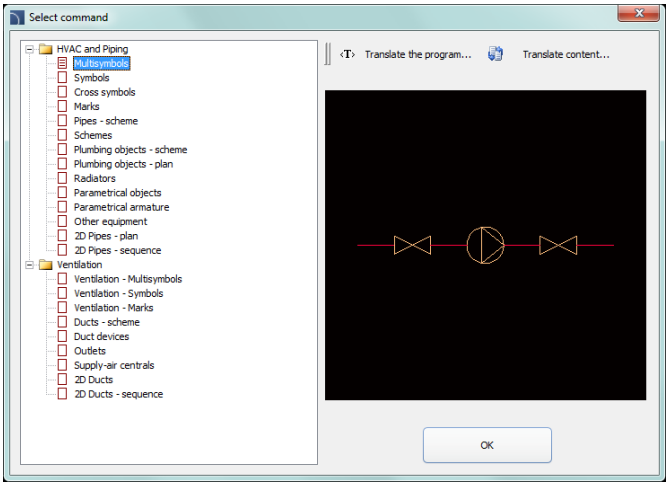
CADprofi HVAC & Piping

HVAC & Piping program helps with designing installations in both general and industrial construction. With many types of predefined and editable lines, this module allows you to design any type of installations such as: heating, water & sewage, gas, medical, fire safety, ventilation, air conditioning, refrigeration and other technical installations in construction and industry. Particularly important for users are functions that support the designing of schemes, strand diagrams, isometrics, plans and sections that allow user to use only one tool to create the entire project.

CADprofi HVAC & Piping in 2 minutes

HVAC & Piping general commands

The **HVAC & Piping** command opens the dialog window from which we can choose the appropriate commands from the **HVAC & Piping** module. This command has been prepared for beginners. In the dialog window apart from the list of all commands we can also find a large preview window that shows the outline of the selected function.



HVAC & Piping command dialog window

Procedures

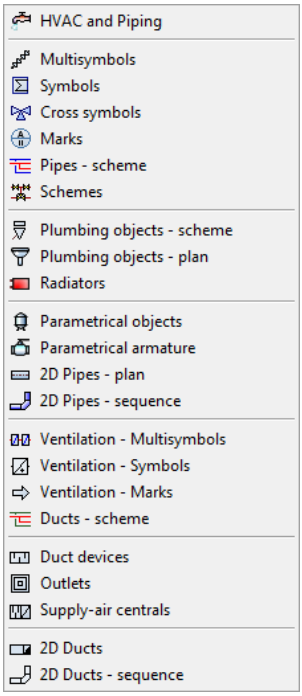
Selecting the command

1. Run the **HVAC & Piping** command.
2. In the dialog window, expand the chosen category and select the needed command.
3. Selected command will start after clicking the **OK** button.

CADprofi HVAC & Piping menu and toolbar

CADprofi HVAC & Piping commands

CADprofi HVAC & Piping toolbar



Menu of CADprofi HVAC & Piping module

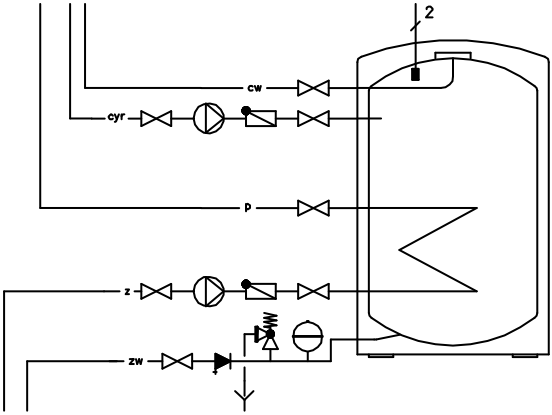
Editing commands

Important editing commands used in designing HVAC and piping installations can be found in the **CADprofi** menu and **CADprofi – Tools** toolbar.

CADprofi – Tools toolbar

CADprofi HVAC & Piping – introduction

Creating schemes is connected with drawing lines, inserting symbols and describing drawings. **CADprofi** program offers the designer appropriate commands that make work easier and faster. Many options automate frequently performed actions, so it's very important to know how to work in the program. An important stage when creating projects is to describe the scheme elements, making it possible to create BOMs of symbols and lines used in projects.



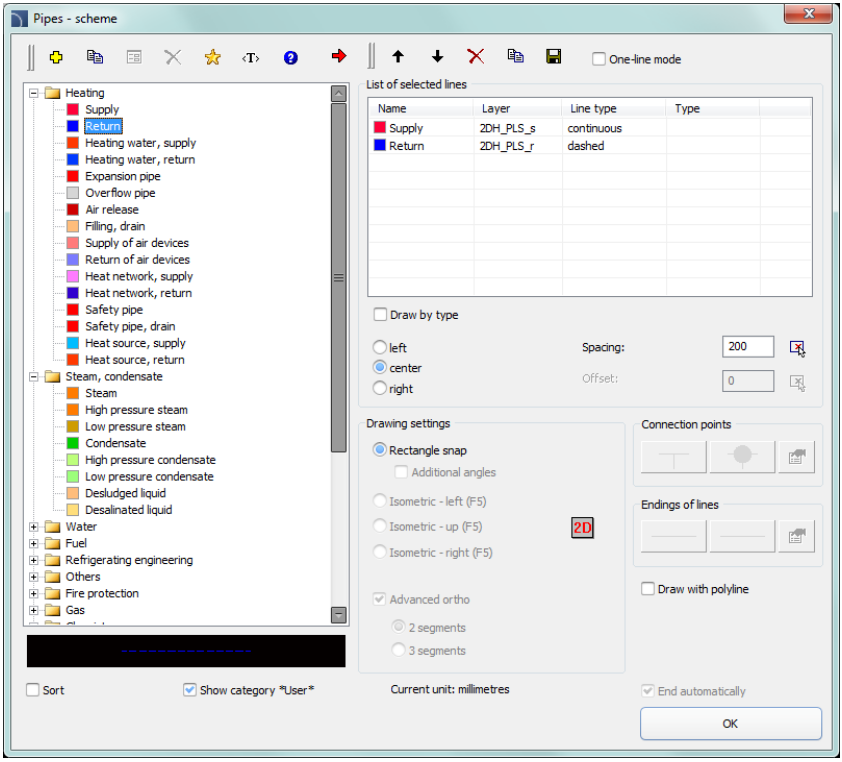
CADprofi HVAC & Piping – main program commands

Schematic lines - general rules

In **CADprofi** program schematic lines are being drawn with lines or polylines which are located on appropriate layers. The term "**line kind**" means the line or specific system purpose (such as **supply line** in a heating installation or a **supply air** line in the ventilation system). Lines may also be given a **type**, which defines their physical properties (e.g. **DN15** for steel pipes, **Cu15x1** for copper pipes etc.). Line type can be specified during the drawing or at any time after user has drawn the lines (with the **Schematic lines – edit** command).
Standard kinds of lines are grouped into appropriate categories. User also has the possibility to define additional kinds of lines.

Pipes/Ducts – scheme

Both commands - **Pipes - scheme** and **Ducts - scheme** makes it possible to draw one or more schematic lines. The process of drawing a single line is possible after selecting in the dialog window the desired line and clicking the **OK** button. Doing so however does not allow to change drawing settings. These options are available only when adding line to the List of selected lines, which is also used for drawing multiple lines.



Pipes – scheme dialog window

The **Pipes – scheme** dialog window contains the following elements:

Lines menu – tools menu that allows user to manage lines' kinds:

- **Define new** – allows to add a new position (line) to the list as a user line. Afterwards it opens a new **edit** window and allows user to edit the line data
- **Copy** – copies the selected line with all its parameters, therefore creating an element made by user. Copied lines may be edited.

- **Edit** – opens the edit properties window of user created lines. Line name can be edited directly by pressing the **F2** key.
- **Delete** – deletes the selected line from the list. Only the lines created by user may be deleted.
- **Add to favourites** – adds the selected line to the ***Favourites*** list.
- **Add element to a list** – adds the selected line to the **List of selected lines**.

List of selected lines – contains lines, which will be drawn after clicking the **OK** button.

Menu of selected lines list – allows to manage the content of the selected lines list.

- **Move selected rows down** – moves the selected line one position down.
  Move selected rows up – moves the selected line one position up.
- **Remove selected rows** – deletes the selected line from the list.
- **Copy selected rows** – copies the selected line and adds it to the list.
- **Save as set of lines** – allows to save the current list of selected lines as a new set.

> ***Notice***
>
> ***It's not possible to create (save) a set of lines if the list contains at least one line from the *Installations from drawing****.

- **One-line mode –** this option blocks the possibility of adding more than one line to the **List of selected lines.** When enabled, program will convert the list of lines to a one line, chosen by user.

Type of line – it is possible to define **Type** for each line.

List of selected lines

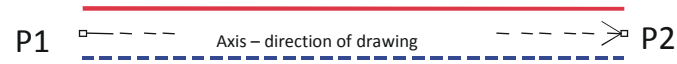
Name	Layer	Line type	Type
Supply	2DH_PLS_s	continuous	
Return	2DH_PLS_r	dashed	
Heating wate...	2DH_PLS_hws	continuous	
Cold water	2DS_PLS_cw	continuous	

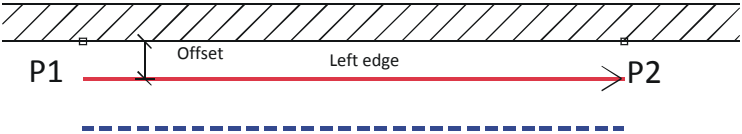
In order to do this, user has to enter an appropriate value in the **Type** field that is located next to the line name on the **List of selected lines**.

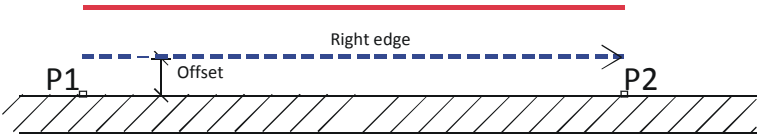
Draw by type – enables the **line type** view for the newly created lines.

Lines conducting options – options that allow user to specify the way of drawing lines.

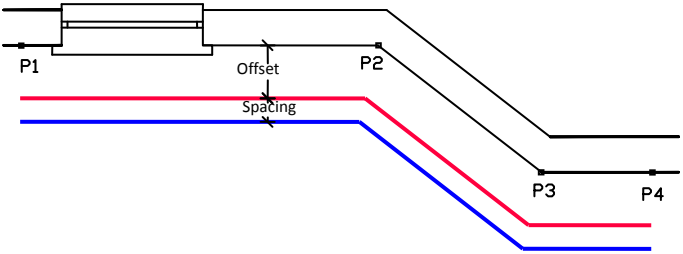
- **Left** – draws lines on the left side of the indicated points.
- **Center** – draws lines symmetrically regarding the indicated points.
- **Right** – draws lines on the right side of the indicated points.
- **Spacing** – the distance between the lines.
- **Offset** – an offset of the drawn lines from the points that are actually indicated in the drawing. This option allows to easily draw lines at a specified distance along the walls or other elements that exist in the drawing.







> ***Hint***
>
> ***Thanks to combining both the drawing edge and the offset user can lead the lines in accordance with any existing objects that are already inserted on the drawing e.g. walls.***



Show category *User* – enables/disables the visibility of the folder that contains all categories and lines created by user. If you turn off this feature then lines defined by user will be "mixed" with other standard **CADprofi** lines.

Drawing settings – line drawing options.

- **Rectangle snap** – 2D drawing default settings.
- **Additional angles** – enables the diagonal dimetric mode, through the use of polar tracking options (POLAR) according to the angle value that was defined in the **CADprofi – options** window.
- **Isometric** – allows to turn on the mode that allows to create isometric schemes (SNAPSTYLE variable). Three options are available (left, up, right) which can be easily switched during drawing isometry by pressing the **F5** key.

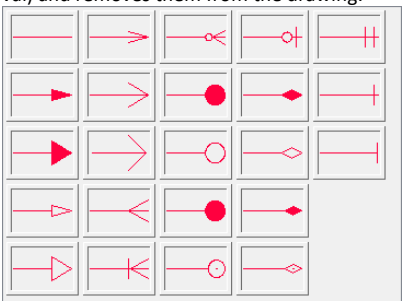
Connection points – when drawing a single line it is possible to automatically insert connection points. There are several options available:

- **Lack** – no symbol on the line joint.
- **Always** – adding a symbol regardless of the kind of connected lines.
- **On the same layer** – adding a connection symbol only when both lines are on the same layer.

> ***Hint***
>
> ***If user uses an extended layer structure this option will add a connection symbol only on lines with the same type.***

- **Connection symbol** – allows to select the graphical symbol that will be inserted in the place where lines connect. User can select one of the following symbols:

- **Settings** – setting up the **scale** (symbols' size) and the **Remove unnecessary connection points** option, which automatically searches for unnecessary connection points (e.g. remnant after line removal) and removes them from the drawing.
- **Ending of lines** – allows user to select symbols that will be inserted in the beginning and ending of lines.

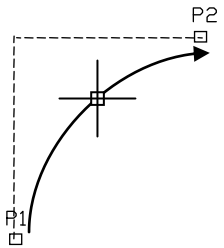


- **Settings** – setting up the **scale** (symbols' size) and the **Remove unnecessary connection points** option, which automatically searches for unnecessary symbols of line endings (e.g. remnant after line removal) and removes them from the drawing.

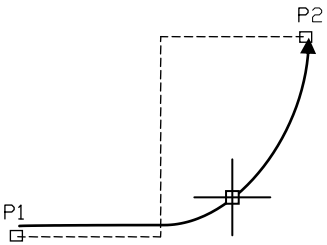
Advanced Ortho – in this mode between the indicated points, two or three orthogonal segments are being drawn. This mode automates the process of connecting objects or lines while maintaining a right angle between the segments.

- **2 segments** – creates a line consisting of two segments.
- **3 segments** – creates a line consisting of three segments.

> ***Notice***
>
> ***Advanced ortho mode is available only in certain CAD programs.***
>
> ***The first drawn segment can be vertical or horizontal depending on which way of the mouse movement is performed after selecting the first point (P1) of the drawn line.***



Advanced ortho 2 segments

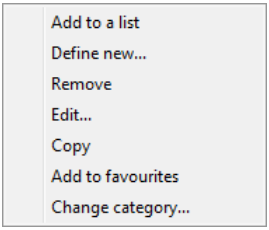


Advanced ortho 3 segments

Draw with polyline – enables/disables drawing line segments as polyline.

End automatically – enables/disables automatic ending of lines when the drawn line is being connected to a different object in the drawing.

Pop-up menu – options menu for selected element/category which is available by clicking the right mouse button. Commands included in the menu are similar to the commands available on the dialog window toolbars.



Procedures

Drawing schematic lines

1. Run the **Pipes - scheme** or the **Ducts - scheme** command.
2. (Optional) Select the line and click the **OK** button in order to allow the drawing of a single line without making changes in the settings.
3. In the newly opened dialog window pull-down the category that contains the desired line kind.
4. Select the desired line and click the **Add element to a list** button, or double click on it in order to put it on the **List of selected lines**.
5. (Optional) Repeat steps 2 and 3 in order to add more lines to the list.
6. (Optional) Using the **Copy selected rows**, **Remove selected rows** buttons add or remove the selected line from the list.
7. (Optional) Using the **Move selected rows down**/**Move selected rows up** buttons change the order of the lines on the list.
8. (Optional) Fill the information about type of line, enable/disable the **Draw by type option**.
9. (Optional) If there is more than one line on the **List of selected lines** specify the **Offset** value.
10. Specify the way how you will draw the line and specify the **Spacing value** if needed.
11. (Optional) If on the **List of selected lines** there is only one object then change the **Drawing settings**.
12. (Optional) If on the **List of selected lines** there is only one object you can set the **connection** and **line endings** options.
13. Enable/disable the **End automatically** option.
14. Click the **OK** button in order to start drawing a line (or several lines).
15. Specify the starting point and next segment points that you want to draw. In order to finish click the **Enter** key.

Pipes/Ducts – scheme

Notice

Offset, additional angles, adding connection points, endings of lines, advanced ortho and end automatically options are available only when a single line is selected.

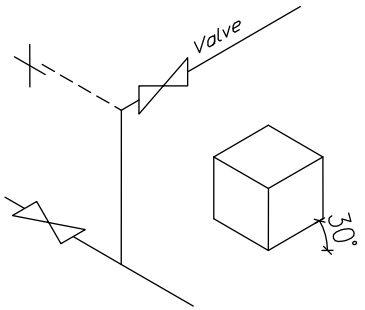
Schematic lines are simple lines or polylines that's why it is also possible to draw lines with the CAD program standard commands such as **Line** or **Polyline**. However, it should be noted, that lines drawn in this way should be placed on appropriate layers. In order to match properties (e.g. layers), between the drawn objects, the **Match properties** (_MATCHPROP) command located in the CAD program can be used.

> ***Hint:***
>
> ***The „Schematic lines – edit" command is used to edit type of lines*** ***(check page 67)******.***

Lines in isometric drawings

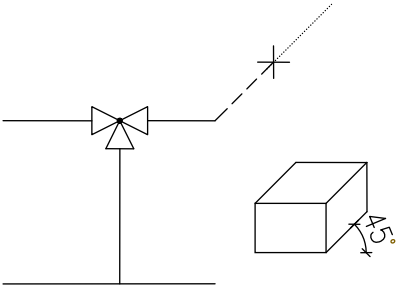
In the **Drawing settings** section user can select optimal CAD settings for easy creation of typical (2D) drawings or different types of axometrical drawings. Isometric mode is only available if only one line has been chosen.

In many CAD programs a special pitch and grid mode is available in which the graphical cursor is adapted to create isometry in which axes are positioned at a 30°, 150° and 270° angle. To enable this mode, when drawing lines select the **Isometric** option in the **Pipes - scheme** or the **Ducts - scheme** dialog window with activating the upper, left or right plane. In isometry lines should be drawn with an enabled **ORTHO** mode. In order to quickly switch to another plane just press the **F5** key.



Drawing isometry

Another possibility to create an isometric drawing is to use the **Additional angles** option, which makes it easier to draw with specific angles. After choosing the **Rectangle Snap** and **Additional angles** options, the **polar tracking** mode is being activated which is used to create drawings in diagonal dimetry.



Drawing dimetry in the polar tracking mode

There is a group of programs based on the IntelliCAD technology in which polar tracking is not available, that's why during drawing there is an option **Cursor obliquity** available, which is used to change the angle of the graphical cursor according to the current dimetry angle. In order to choose this option in the command line, user should enter the "**O**" and press **Enter**. Cursor will change the incline angle. In order to return to the normal position user should choose the **Cursor obliquity** option again.

Example text in the command line:

```
Specify first point or [cursorObliquity]:
Specify next point or [Undo/cursorObliquity]:
Specify next point or [Close/Undo/cursorObliquity]:
```

Dimetrical angle is defined in the **CADprofi - Options** (check page 30) dialog window. When drawing lines in dimetrical mode user should use the ORTO mode, which can be turned on and turned off with the **F8** key.

Notice

In order to select an option in the command line user should type a letter or several letters, which are written in the option name in capital letters. In the above example to choose the Undo command user should type a "U" letter and after that press the Enter key. Instead of the Enter key, user can press the right mouse button or the spacebar.

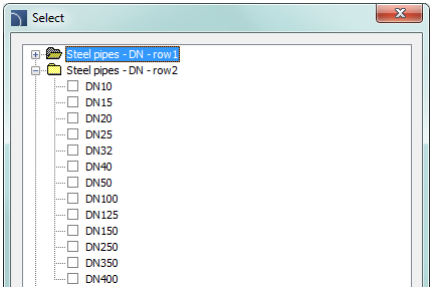
Isometric drawings

Drawing risers (vertical lines)

In schematic lines it is possible to present vertical lines segments using special symbols, such as **risers**.

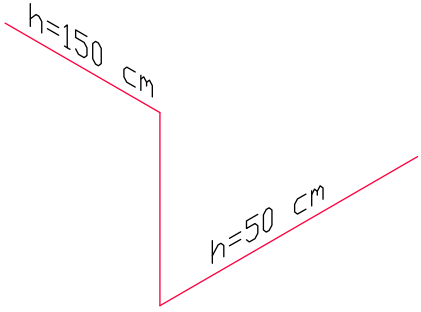
Despite the fact that **risers** are being inserted using the **Symbols** (check page 172) command, program treats them as **schematic lines**.

For risers user should specify the type of line and height (length of vertical line segment). Line type can be selected from the list that will be available after pressing the **Browse** ... button. Thanks to this, their diameter data and length are being included in BIMs.

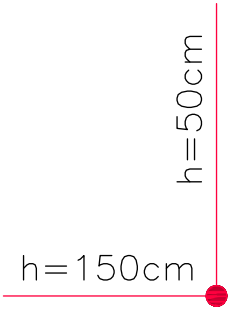


Data saved in the risers' blocks is included when creating a BIM with use of **Plumbing/heating - lines** (check page 57).

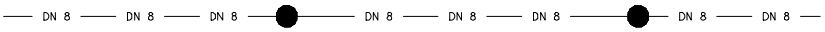
Below example shows the use of risers' symbols in the project.



The actual installation view



Schematic view



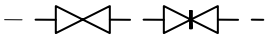
Scheme with risers view

Name	Line type	Sum	--
Line	DN8	2.37	m
Line	DN8	1.23	m
Line	DN8	1.96	m
Line	DN8	1.00	m
Line	DN8	2.20	m

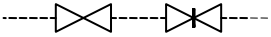
Lines BOM view

Linetype scale for lines

Each line is being drawn with different line patterns, such as a dashed line (DASHED) used for the "Return" line. The line pattern density is set using the CAD command: LTSCALE (_LTSCALE). By default new drawings have got a linetype scale factor set as 1.00. If this factor number is too small or too big then line pattern may not be visible and all lines will be seen as drawn with a continuous line.



LTSCALE = 200



LTSCALE = 50

In **CADprofi** program it's possible to design in units set as millimeters, centimeters or meters. For this reason, when drawing schemes the linetype scale should be always adapted to the edited scheme. For example, when drawing a scheme with the unit set as "millimeters" with symbol's size set as 200 then the linetype scale factor should be between the 50 and 200 range.

Defining new user lines

User can add his own type of lines to the program. In this way, program can be adapted to user's needs and can be used in a variety of projects in which it is necessary to draw schemes.

Procedures

Adding new lines in the database

1. Run the **Pipes - scheme** or the **Ducts – scheme** command.
2. Click the **Define new** button in order to add a new line to the existing database. New element will be added to the ***User*** category. A new **Edit** window will open, in which user will be able to specify all line parameters.
3. (Optional) Select an existing line and click the **Copy** button to create a new item. Selected line will be used as a template.
4. Change the new line settings in accordance with the **Edit line data** procedure.

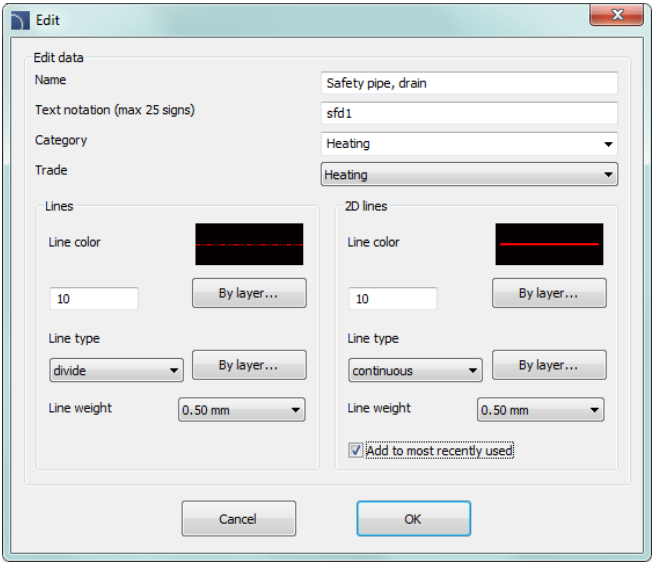
***Favourites* category**

It is also possible to create a folder that contains a list of most frequently used installations by the user. This folder is created automatically when a first element marked as **favorite** has been added to the program. In the ***favorite***category user can define any branch structure, in which the individual elements will be located.

Edit line data

CADprofi allows to edit parameters (drawing features) of lines created by user. To do this, use the **Edit** option.

In **CADprofi** program pipe and ventilation lines can be drawn as schematic (with use of lines or polylines) or in 2D view. Therefore, when editing properties it is possible to specify different parameters for schematic lines as well as for lines in 2D view.



The Edit line data dialog window

The **Edit** dialog window contains the following items:

Name – name of edited line.

Text notation – text marking for lines used as an ending of layer's name in the extended layer structure.

> ***Notice***
>
> ***It is recommended to use „unique values" of text notations in order to ensure proper recognition of lines by CADprofi.***

Category – currently assigned category in **CADprofi** database.

Branch – currently assigned industry branch in **CADprofi** database.

Add to most recently used – this option adds the edited line into the default list of line types in the in the **2D Pipes – plan** dialog window.

Lines or **2D lines** – settings used for schematic lines or lines drawn in 2D:

- **Line color** – determines the line color. Color can be selected in several ways: by clicking on the button that contains the line preview, with the help of the **by layer** option or by entering CAD program color number.
- **Line type** – determines the line type (kind). Line type can be selected from the list of available line types or from the **by layer** option.
- **Line weight** – determines the line width. Line width can be chosen from the list available values in the program.

> ***Notice***
>
> ***Only lines defined by user may be edited (it is not possible to change line parameters from the primary CADprofi database).***

Procedures

Edit user lines

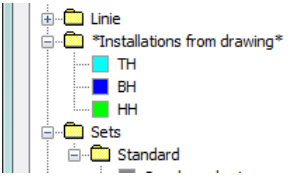
1. Run the **Pipes - scheme** or **Ducts - scheme** command.
2. (Optional) If the line that you would like to edit is located on the list of selected lines, then select it and click the **Remove selected rows** button in order to remove it from the list.
3. On the **List of available lines** select the line that you would like to edit.
4. Click the **Edit** button A new dialog window will open, allowing you to edit the line.
5. (Optional) Change the **Name** of edited line.
6. (Optional) Change the **Text notation** of edited line (not recommended if the edited line is already inserted in the drawing).
7. (Optional) Change **Category** of edited line.
8. (Optional) Change the **Trade** for selected line.
9. (Optional) Enable/disable the **Add to most recently used** option.
10. (Optional) Change parameters of schematical **Lines**.
11. (Optional) Change parameters of schematical **2D Lines**.
12. Click the **OK** button to accept the changes.

> ***Notice***
>
> ***If you have changed the parameters for lines that have already been added to the drawing then in order to update their properties you will need to insert a new line (that has the new parameters) into the drawing.***

Installations from drawing

All lines and polylines that lie on appropriate layers are treated by **CADprofi** program as schematic lines. This includes layers with names similar to **2DX_PLS_xxx**.

When several designers are working on the same project, then it is possible that one of these designers has used his own line kinds and then has forwarded the drawing to further edition. In this case, **CADprofi** application will automatically detect the presence of additional lines (installation kinds) and will display them in a new category called **Installations from drawing**. Name of the new lines that were identified from the drawing are being taken from their label (ending of the layer's name). If these lines' parameters will be edited in the dialog window, then they will be permanently saved in the program's database as user's lines.



It is possible to use **Installations from drawing** without the need of editing them, but in this case, on all BOMs the **Installations from drawing** lines will have their names listed as their layers' label.

Lines' layers

Each line kind is being drawn on separate layers. In order to easily manage layers in the top menu of **CADprofi** programs are located additional commands that allow to quickly hide or show all layers that belongs to central heating, sewage, water and other kinds of installations.

In the *C:\Cadprofi\X.x\PlotStyle* printing styles are located that suit the program's layers structure. Out of these files user can select monochrome printing (*CADprofi Mono.ctb*) and color printing (*CADprofi Color.ctb*).

*.ctb files should be copied to the appropriate folder of used CAD program.

For more information, please refer to the layers section (check page 32).

Working with symbols – general rules

CADprofi program possesses several commands used for inserting symbols into the drawing. It is possible to insert symbols in any point, into lines, as well as the simultaneous insertion of multiple symbols in the drawing.

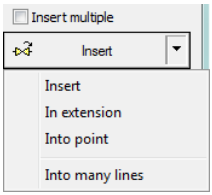
CADprofi symbols have certain properties, thank to which they behave in a "smart" way, for example automatically adjust the insertion angle to the line, automatically break the line at insertion point, adjust the layer depending on the kind of installation etc.

All symbols available in **CADprofi** program are being inserted as blocks, which can be copied, deleted or edited with the use of standard CAD program commands. Most symbols can be also edited with the use of **CADprofi** editing commands such as: **Quick edit**, **Edit symbols** and **Delete symbols**.

Symbols are often used to graphically represent real devices, armature and other installations elements, therefore it is possible to add technical parameters, about the products used in the project, to the symbols (**Attributes and descriptions** command), as well as numbering and symbols' marks. This information can be used when describing drawings when creating graphical **legends or specifications** (check page 46).

The following commands are used in order to insert symbols into the drawing: **Multisymbols**, **Symbols** and **Marks**.

Each command that is used for working with symbols has got a variety of options used insertion of single or multiple symbols into the drawing.



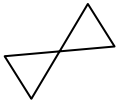
Main options (available after pressing the button):

- Insert symbol or symbols.
- Into many lines.
- In extension.
- Into point.

'Insert symbol' option

This is the default option that allows to insert symbols in any point or into a line indicated by user.

Insert into a line



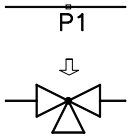
Insert in any point

Procedures

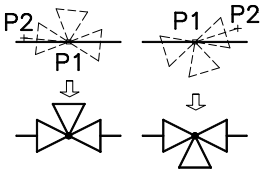
Insert symbol

1. Select the symbols or symbols in any library.
2. Click the **Insert symbol** button. The library dialog window will close allowing insertion of symbols into the drawing.
3. Point into a line, in which you would like to insert the symbol.
4. (Optional) Click anywhere in the drawing to insert a symbol.
5. (Optional) Specify the rotation angle of inserted symbol - this option is available only if user has selected the **Rotate** option in the library dialog window.

When inserting symbols into lines, the symbol insertion angle depends on the line angle (the angle specified in the dialog box is being ignored).
In this case, if user selects the **Rotate** option then he will be able to specify symbol's rotation relatively to the line. This option is very convenient when inserting symbols which have a possibility to determine their tap direction, handle position, flow direction etc.
The angle of such symbol will be dependant on the position of P2 point.



Without rotation

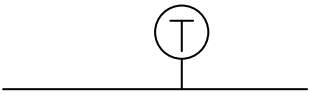


With rotation

'Into point' option

The **Into point option** may be used when it is necessary to insert the symbol in a point that lies on a line, but in such a way that the symbol won't "adjust itself to the line". In this option, the inserted symbol "ignores" the line, so it does not break it and it does not take angle from it. This option is used in many situations for example when inserting sensors, which usually need to touch lines.

Inserting symbol in a line point



Inserting sensor in a line point

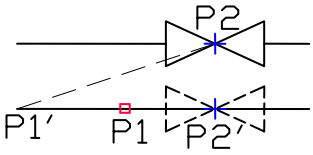
Procedures

Insert into indicated point

1. Select the symbols or symbols in any library.
2. With the use of button, pull-down the insertion list.
3. Click the **Into point** button. The library dialog window will close allowing insertion of symbols into the drawing.
4. Point into a point in a line, in which you would like to insert the symbol.
5. (Optional) Click anywhere in the drawing to insert a symbol.
6. (Optional) Specify the rotation angle of inserted symbol - this option is available only if user has selected the **Rotate** option in the library dialog window.

'In extension' option

The **In extension** option allows to precisely specify the symbol insertion point in relation to other objects located in the drawing. This allows to insert a symbol in a point in accordance with for example another symbol.



Inserting in extension

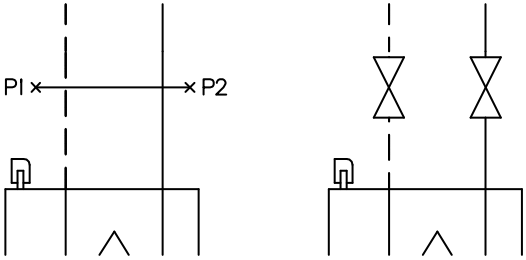
Procedures

Insert in extension

1. Select the symbols or symbols in any library.
2. With the use of button, pull-down the insertion list.
3. Click the **In extension** button. The library dialog window will close allowing insertion of symbols into the drawing.
4. Click a line, in which you would like to insert the symbol (**P1**).
5. Specify the symbol insertion point against you would like to specify the location of the inserted element (**P2**).
6. Specify an additional offset or click **Enter** to confirm the insertion point.
7. (Optional) Specify the rotation angle of inserted symbol - this option is available only if user has selected the **Rotate** option in the library dialog window.
8. Symbol will be inserted in the drawing and the command will end (if user has not selected the **Insert multiple** option).

'Into many lines' option

Into many lines option gives the possibility to quickly insert selected symbol into many lines. It allows to insert selected symbol into all lines that cross with the „indication line", that was specified during insertion (**P1**–**P2**).



Inserting symbols into many lines

Procedures

Insert into many lines

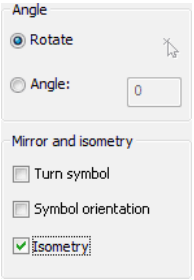
1. Select the symbols or symbols in any library.
2. With the use of button, pull-down the insertion list.
3. Click the **Into many lines** button. The library dialog window will close allowing insertion of symbols into the drawing.
4. Specify the first point of the "crossing line" that will cross with lines (**P1**).
5. Specify the second point of the "crossing line" that will cross with lines (**P2**).
6. The selected symbol will be inserted in all lines that crossed with the "crossing line".

Symbols in isometry

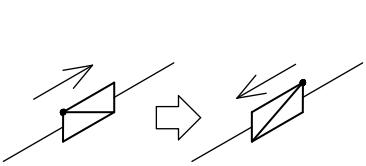
CADprofi allows to create axonometric drawings. **CADprofi** application performs an appropriate transformation of inserted symbols in such a way that they are given an inclination that corresponds with the current isometric angle.

After selecting the **Isometry** option, the meaning of rotation and mirror controls changes. The default symbols insertion angle is the isometric angle that is defined in the **CADprofi Options** (check page 30).

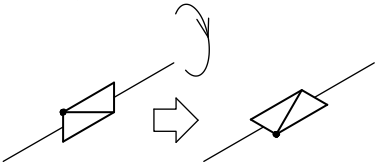
- **Turn symbol** allows to insert a reverse symbol, e.g. directed in opposite direction.
- **Symbol orientation** allows to insert a symbol that is rotated with respect to the selected isometry axis.



Symbol orientation may differ depending on the isometry axis. This means that with the same settings, the symbol inserted in the right axis can be oriented horizontally, and if is inserted in the left axis can be oriented vertically. Therefore, user will need to gain experience in working with **CADprofi** in order to create axometrical drawings efficiently.



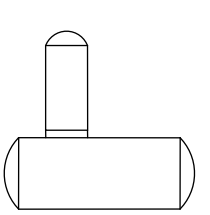
Turn symbol



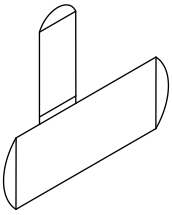
Symbol orientation

Isometric views for symbols are being generated automatically. Some symbols were not intended to be used in axonometric drawings. Therefore, for a small number of symbols, their isometric view may be generated in an invalid way.

User may add his **own new symbols** (check page 89) to **CADprofi**. These symbols can also be used in axonometric drawings. Simply add a normal symbol to the **CADprofi** application and isometric views of these symbols will be created automatically by the program. When defining your own blocks, user should avoid using complex objects, such as multilines, splines, nested blocks, hatches etc.



User symbol



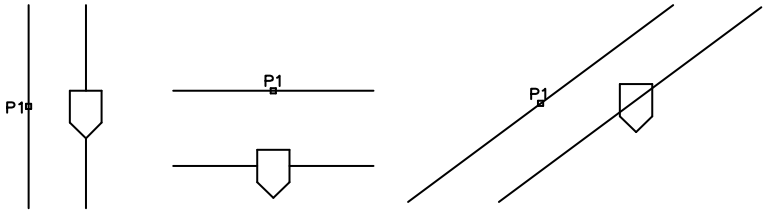
Automatic isometric view

Additional information about creating isometric drawings is available in the **Pipes/Ducts – scheme** (page 148) and **Attributes and descriptions** (page 34) chapters.

Usage of symbols inserted with a fixed angle

In order to maintain the project correctness, some symbols are being inserted with a fixed angle. Examples include gravels and separators. These symbols should be inserted with a 0° angle.

In **CADprofi** many symbols have got an enforced insertion angle. When user will insert such symbols in horizontal or vertical lines, these lines will automatically break. In other cases, lines are not being broken.



Vertical line *Horizontal line* *Incline line*

Notice

For symbols with a fixed insertion angle, "Angle" and "Rotation" options located in the dialog box are being ignored.

Symbols that contain text (visible attributes)

For symbols that contain text (visible attributes) **CADprofi** program will automatically adjust angle of the text. Thanks to this the situation in which text would be displayed "upside down" or "backwards" is being avoided. User has the following possibilities to adjust texts:

- **0° fixed angle**.
- **0° or 90° angle**. If symbol is inserted with a 90° angle, then the text is also rotated by 90°. In other cases the text angle is 0°.
- **Automatic angle** from -30° to +120°. In these ranges the text rotation angle is sufficiently readable.

Adjustment of attributes angle

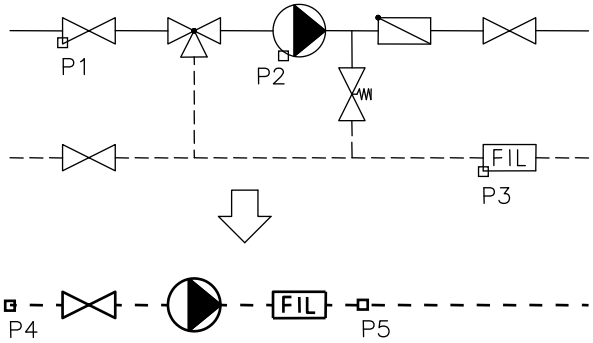
Notice

Options for text adjusting are permanently set for each symbol. User can change these options only for symbols that were added by the user.

Smart copying of symbols

In order to accelerate the work, **CADprofi** program allows for smart copying of symbols from the current drawing.

The **Pick symbol** option is available in the **Multisymbols** and **Ventilation - Multisymbols**. After selecting this option, program will hide the dialog window and enable selection of symbols that will be copied (e.g. **P1**, **P2**, **P3**). Symbols will be copied in the same order that they were selected in the drawing. It is possible to select a single symbol repeatedly. After the selection of symbols is done, press the **Enter** key (or right mouse button). The dialog window will be displayed again, in which user will be able to freely specify all options (check page 177) related to insertion of copied symbols. The process of inserting these symbols is the same as for the symbols selected from **CADprofi** libraries (e.g. inserting into a point or in a line – **P4**, **P5**)



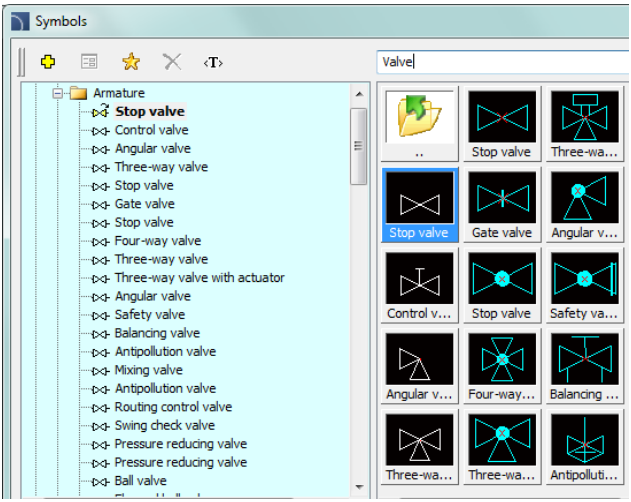
Copying of three symbols with inserting them in a line

Notice

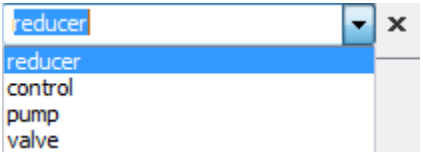
If during the symbol selection, user will select an object, which is not compatible with the command, then it won't be added to the list of copied symbols. It is also unacceptable to simultaneously copy symbols from different modules such as for example electrical and piping symbols.

Find symbols

In each dialog window that is used for symbols insertion there is a search option, thanks to which it is possible to quickly find desired symbols.



The **Find** field is used to enter a phrase (full or just a part name of searched symbol). This field remembers typed words, which allow user to search again from the expanded list by pressing the ▼ button and clicking **Find**.



After filling the search field, application will automatically filter the current library content and display only those categories which contain the objects that match the search results. Clicking the **Restore** button ✖ will display all library objects and end the search.

Multivariant symbols

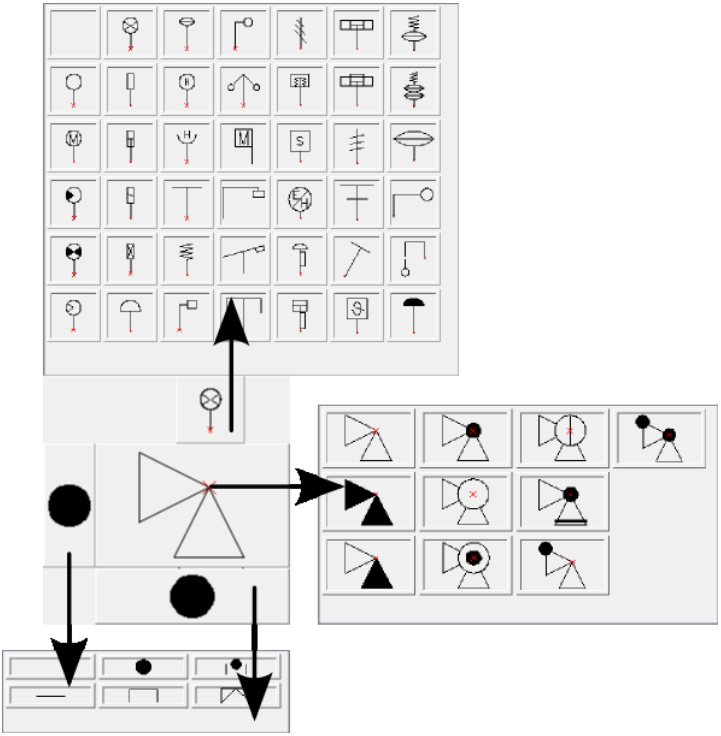
Multivariant symbols are objects, thanks to which in a quick and easy way user can customize their view or information contained in the inserted object. In this way, one multivariant symbol can replace many "traditional" ones, while giving a better possibility to configure their look and type.

Multivariant symbols may be freely modified, using the **Edit symbols** command.

Example of a multivariant symbol

Angular valve – the symbol options allow user to specify:

- type of valve,
- type of propulsion,
- type of connection regardless of both endings.



Management of symbols content

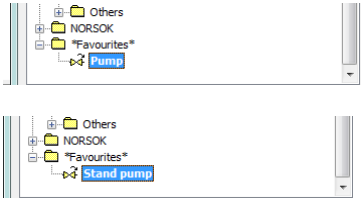
CADprofi program possesses several thousands of symbols grouped in appropriate categories. Witch such a large number it is sometimes hard to find symbols needed at the moment. On the other hand, when working on a particular project it is usually not needed to work with more than several dozens of symbols. In order to speed the work, user can add selected items to the ***Favourites*** category thanks to which he will be able to get a much faster access to them.
It is possible to create sub-categories related to a specific theme, such as a separate category for symbols used in designing boilers, sanitary installations and more. Furthermore, one element can be added to many sub-categories.
For elements placed in the ***Favourites*** category it is possible to change their name, which will be later used in BOMs, legends and descriptions. Change of name can be also used to user own translations for some symbols from international standards.

Adding symbols to favourites

Add to favourites button allows to copy any symbol to the ***Favourites*** category. This folder is created automatically after user has copied the first symbol.

Changing the symbol name

Initially, each copied symbol receives the default name from the original item.
In order to change the symbol name that is located in the ***Favourites*** folder, user should first select it and then press the **F2** button to activate the edition mode.

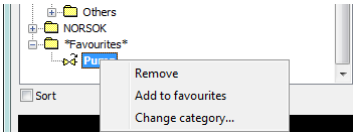


Notice

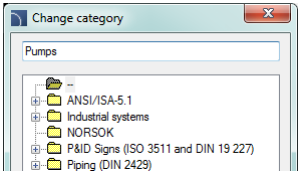
Changed name will be visible in tables and descriptions that are created with the „Bill of materials” (check page 46) command.

Changing category

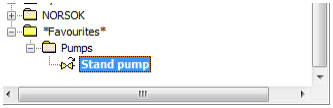
In order to change or define a new category, click the right mouse button on a desired symbol and from the context menu choose the **Change category** option.



In the dialog box, user can select an existing category or enter a new category name. After accepting, selected symbol will be moved to the defined category.

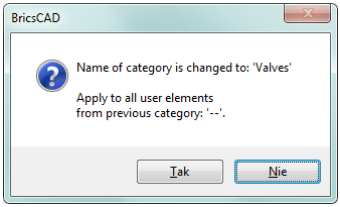


CADprofi application allows to create many copies of each symbol and it is possible to place them in different categories. Thanks to this user can create categories that group symbols of a specified type or the most common used symbols for a given industry type.



Changing the category or many symbols

If user is editing a symbol in his native category in which there are many symbols, then **CADprofi** application will change category for all these symbols.



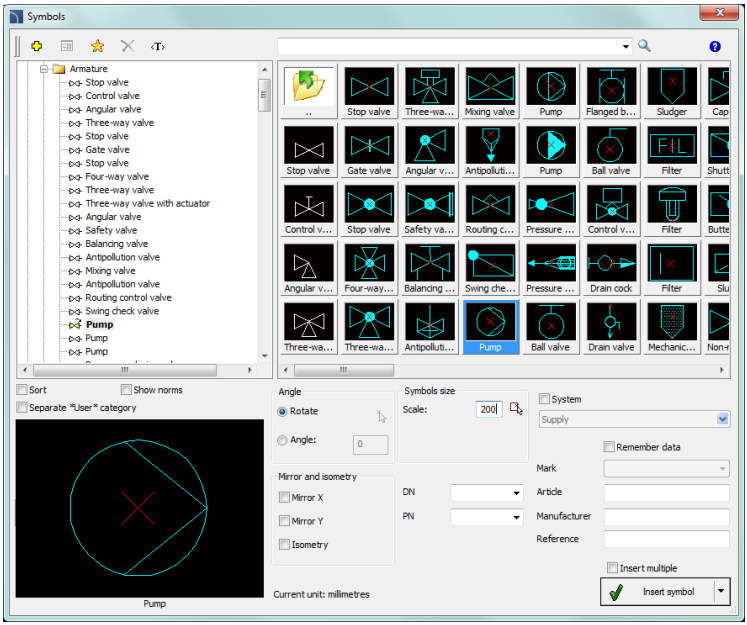
Deleting symbols

It is possible to delete symbols from the ***Favourites*** category. To do so, select the symbol and choose the **Delete** option from the dialog window.

Symbols

Both **Symbols** and **Ventilation - Symbols** commands allow to insert various symbols from the available library.

When inserting symbols into a line, application will automatically break the line in the insertion point, while maintaining the rules of creating schematic drawings.



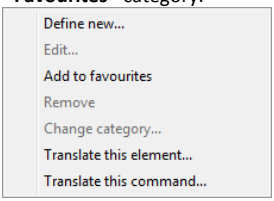
Symbols dialog window

The **Symbols** dialog window contains the following elements:

Tools menu:

- **Define new** – allows to add a new element to the database. Read more about it in the Defining user blocks (check page 89).
- **Edit** – allows to Editing user block parameters (check page 92).
- **Add to favourites** – copies symbol to the *Favourites* (check page 170) category.
- **Delete** – deletes the selected symbols from the ***Favourites*** category.

Pop-up menu– options menu for the selected symbol, which is available after right clicking on the mouse button on the selected symbol. Commands in this menu are similar to the tools menu.



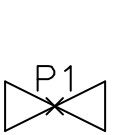
Categories tree – a set of all categories and symbols in the library, shown as a pull-down tree. User can customize the display order by using the following options:

- **Sort** – alphabetical order of categories placement and their content.
- **Show norms**– enables/disables displaying symbols according to norms.

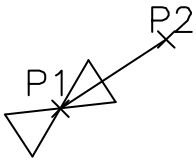
Thumbnail view – displays the content of currently selected category as thumbnails.

Preview – displays the preview of the currently selected symbol. Right-clicking on the preview window will zoom the view.

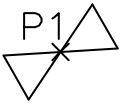
Angle/Rotation – possibility to specify a fixed angle when inserting symbols or enabling the rotation option, in which symbol's angle is defined by rotation during insertion. During insertion of symbols into lines, the insertion angle depends on the line angle (the angle specified in the dialog window is being ignored).
When inserting symbols into lines it is possible to use the **Rotation** option in order to dynamically specify symbol rotation relatively with the line. This option is very convenient when inserting symbols of pumps, vales etc.



0° angle



Rotation



30° angle

Inserting into a line

Scale – specifies the size of inserted symbols. It is possible to enter the scale value or to indicate a symbol on the drawing whose scale user would like to use.

System – if in the program options, the extended layer name structure (check page. 29) is enabled then symbols will be inserted on layers depending on the kind of installation (system). In case of inserting symbols into an existing line then the **System** option is being ignored and the symbol layers depends on the line kind.

Mirror and isometry:

- **Mirror X, Y**– enabling this option will create an X or Y mirror image for the specified object.
- **Isometry** – enabling this option will convert the symbol, so it will possible to use it in isometrical schemes.

- This option may not be available for some symbols.

Object data– this option allows to specify symbols technical or identification parameters. For each symbol, different parameters can be given such as: **Article**, **Manufacturer**, **Reference** (Catalog number), **DN**, **PN** and others. In order to fill the parameters enter any text or choose a value from the available list.

Remember data– this option allows to save the user entered parameters. This option should be enabled when user inserts symbols with similar characteristics.

Insert multiple– enables/disables the possibility of inserting multiple symbols to the drawing. The insertion process requires from the user to press the **Enter** or **Esc** key when he finished inserting multiple symbols.

Procedures

Inserting symbol

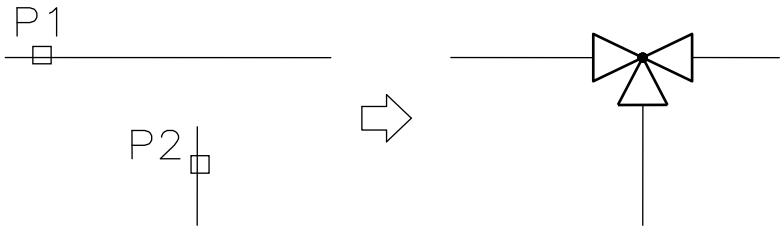
1. Run the **Symbols** or **Ventilation – Symbols** command.
2. In the dialog window choose the appropriate category.
3. Select the symbols that you would like to insert. In the bottom-left corner of the dialog window a preview of the selected symbol will be displayed.
4. Specify the insertion parameters: **System**, **Rotation**, **Scale**.
5. (Optional) In accordance with your preferences fill out the value of following parameters: **Article**, **Manufacturer**, **Reference**, **DN**, **PN**.
6. Click the **Insert symbol** button in order to specify the symbol insertion point.
7. (Optional) With the use of the button pull-down the insertion list and choose **In extension** to specify the insertion point in accordance to other objects.
8. (Optional) With the use of the button pull-down the insertion list and choose **Into point** to insert the symbol in a specific point.
9. (Optional) With the use of the button pull-down the insertion list and choose **Into many lines** to insert symbols simultaneously into many lines.
10. Click on a point in the line to insert the symbol.
11. (Optional) Click on any point in the drawing, to insert the symbol without inserting it on a line.
12. (Optional) Specify the inserted symbol angle - this option is available only if the **Rotation** option in the library dialog window has been enabled.
13. After closing the dialog box, insert the symbol into the drawing.

Symbols

Cross symbols

The **Cross symbols** command is used for automatic insertion of angular, three-way or four-way symbols in line crossings.

The default cross symbol insertion process involves indicating two lines. After that program automatically inserts a symbol in the intersection. Indicated lines can be interrupted or properly extended.



Inserting a cross symbol

The basic functionality and symbols options are being described in the **Symbols** (check page 172) chapter.

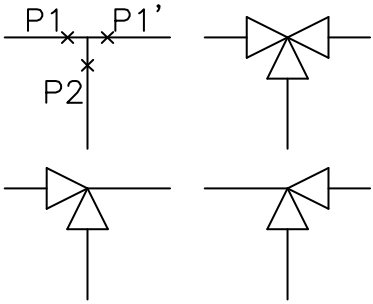
Procedures

Inserting a cross symbol into a line crossing

1. Run the **Cross symbols** command.
2. In the dialog window choose the appropriate category.
3. Select the symbols that you would like to insert in the drawing.
4. Check the scale and symbols parameters.
5. (Optional) In accordance with your preferences fill out the value of following parameters: **Article**, **Manufacturer**, **Reference**, **DN**, **PN**.
6. Click the **Insert symbol** button in order to indicate lines in which you would like to insert selected symbol.
7. Indicate the first line in the drawing in which the symbol should be inserted (**main line P1**).
8. Indicate the second line in the drawing (**P2**).
9. Symbol will be inserted in the drawing, lines will be properly interrupted or extended or shortened.

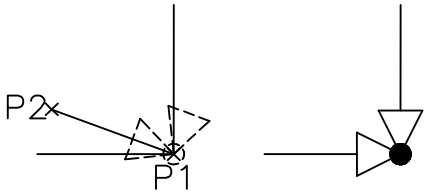
Cross symbols

When inserting three-way symbols the first point (**P1** or **P1'**) defines the main line, while the next point (**P2**) defines a secondary line. In case of an angular valve choosing the point (**P1** or **P1'**) defines the direction in which the valve is being inserted.



Inserting cross symbols

A second possibility of inserting a cross symbol is to indicate the actual line crossing of two lines (**P1**) and afterwards indicate the (**P2**) point, which specifies the symbol rotation. **CADprofi** application will automatically detect and interrupt the connected lines.



Inserting a cross symbol

Notice

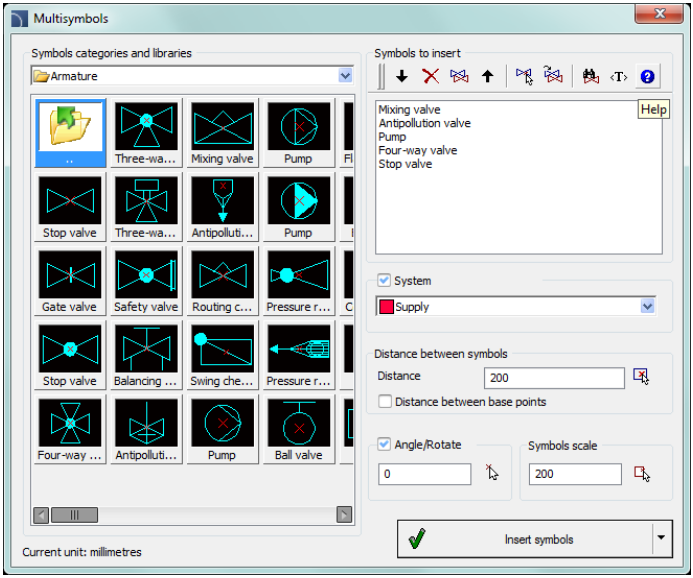
Automatic adjustment and lines interruption takes place if indicated lines intersect with each other at a right angle.

Inserting a symbol at the line crossing is possible only for the "Insert symbol" option (other options, such as "Into many lines" will insert a symbol accordingly to the description in the „Symbols". command).

Inserting a symbol at the line crossing won't be possible if one of these lines is a polyline.

Multisymbols

Multisymbols and **Ventilation – Multisymbols** commands allow simultaneous insertion of many symbols in one row. These symbols may be inserted either in an existing line or in a chosen point. Symbols are being grouped in categories according to their standards or according to their element types.



Multisymbols dialog window

The **Multisymbols** dialog window contains the following items:

Symbols categories and libraries – display a list of library categories and their content. In order to change the category user should expand the list using the button, and then click on the selected item. After that the content of this category will be displayed as thumbnails. Click on a chosen symbol in order to add it to the list of symbols to insert.

Symbols to insert – this window contains a list of symbols, which will be inserted into the drawing, preserving their order. Their order can be changed using the move up, move down buttons and the drag and drop method. Other options:

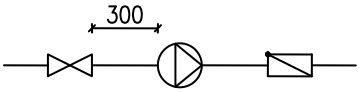
- **Delete** – deletes the selected symbol from the list.
- **Copy** – copies the selected symbol that's located on the list.
- **Pick symbol** – selects a symbols from the drawing and inserts it on the list.
- **Most recently selected symbols** – adds to the "Symbols to insert list", elements from the previously performed **Multisymbols** command.

- **Find** – opens the find symbols dialog window.

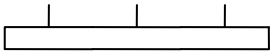
> ***Notice***
>
> ***List of inserted symbols can contain a maximum of 20 positions that can be inserted in one command.***

System – in case of selecting one or more symbols into the drawing a row will be inserted with all the selected symbols. In this row user can also add a line that is specified in the **System** field (e.g. supply, water etc.).
If in the program options, the extended layer name structure (check page. 29) is enabled then symbols will be inserted on layers depending on the kind of installation (system).

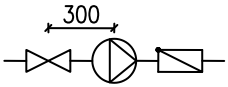
Distance between symbols – specifies the distance between the symbols that are being inserted in a single command. This distance can be measured between two extreme points or between base points of each symbol. Space between the symbols will be filled with a user selected line (**System** option).



Distance between symbols



"0" distance between symbols



Distance between base point

"0" distance between symbols

> ***Hint***
>
> ***If distance value is set as „0", then symbols will be drawn directly one after the other.***

Scale – specifies the size of inserted symbols. It is possible to enter the scale value or to indicate a symbol on the drawing whose scale user would like to use.

Procedures

Pick symbol with appropriate scale.

1. In the library window click the **Pick symbol with appropriate scale** button.
2. Library window will close, allowing user to indicate a symbol in the drawing.
3. After indicating the symbol the dialog window will appear again and the scale value will be set in accordance with the scale of the indicated symbol.

> ***Hint***
>
> ***In CADprofi it is recommended to use an appropriate scale in accordance with the currently used drawing units. For designing in mm the default scale is 200, for cm 20 and for m is 0.2.***

Angle/Rotation – possibility to specify a fixed angle when inserting symbols or enabling the rotation option, in which symbol's angle is defined by rotation during insertion. During insertion of symbols into lines, the insertion angle depends on the line angle (the angle specified in the dialog window is being ignored).
When inserting symbols into lines it is possible to use the **Rotation** option in order to dynamically specify symbol rotation relatively with the line. This option is very convenient when inserting symbols of pumps, vales etc.

Inserting into a line with rotation

Insert symbol –the default option that allows to insert symbols in any point or into a line indicated by user. When inserting symbol into lines, the lines are being automatically interrupted in appropriate points.

Procedures

Inserting multisymbols

1. Run the **Multisymbols** or **Ventilation – Multisymbols** command.
2. In the dialog window choose the appropriate category from the **Symbols categories and libraries**.
3. Click on the symbol thumbnail. It will be added to the **list of symbols to insert**.
4. (Optional) Click on a thumbnail of next symbol that you would like to add to the list.
5. (Optional) Change the category if you would like to add a symbol from a different library.
6. (Optional) Click the **Pick symbol** button. The command window will be hidden and it will be possible to indicate a symbol or symbols that will be inserted in the list of symbols to insert. After indicating the needed symbols click the right mouse button or press the **Enter** key to make the **Multisymbols** window visible again.
7. (Optional) With the use of the drag & drop method or with the use of buttons change the order of symbols on the list.
8. (Optional) With the use of the **Copy**, **Delete** buttons, copy or delete symbols indicated from the list.
9. Specify the insertion parameters: **System**, **Rotation**, **Scale**, **Distance**.
10. Click the **Insert symbol** button in order to specify the multisymbols insertion point.
11. (Optional) With the use of the button pull-down the insertion list and choose **In extension** to specify the insertion point in accordance to other objects.
12. (Optional) With the use of the button pull-down the insertion list and choose **Into point** to insert multisymbols in a specific point.
13. Click on a point in the line to insert the multisymbols.
14. (Optional) Click on any point in the drawing, to insert the multisymbols without inserting it on a line.
15. (Optional) (Optional) Specify the inserted multisymbols angle - this option is available only if the **Rotation** option in the library dialog window has been enabled.
16. Multisymbols will be inserted into the drawing.

Multisymbole

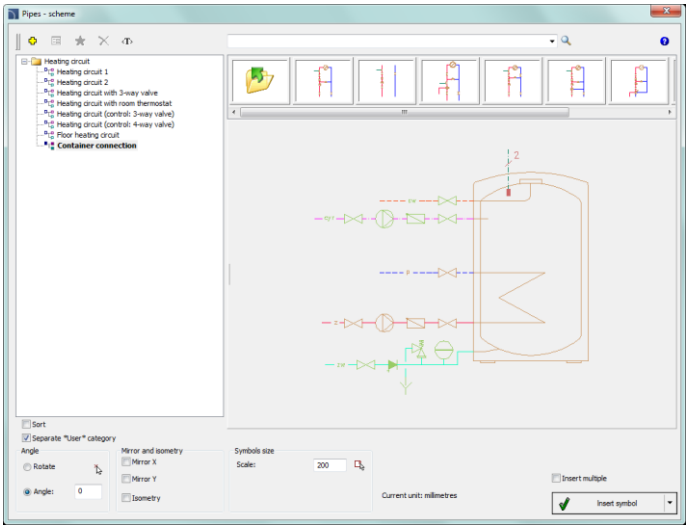
Notice

The„Into many lines" option in „Multisymbols" command is active only if one symbol has been selected.

Schemes

The **Schemes** command allows to insert already prepared schemes or typical schemes fragments such as containers connection systems, various valves systems, heating circuits, typical boiler schemes, heating schemes, etc.

When inserting, scheme is often broken into it parts. Thanks to it is possible to edit the drawing, add descriptions etc.



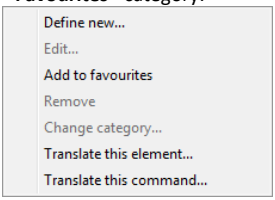
Schemes dialog window

The **Schemes** dialog window contains the following items:

Tools menu:

- **Define new** – allows to add a new element to the database. Read more about it in the Defining user blocks (check page 89).
- **Edit** – allows to change selected user block.
- **Add to favourites** – copies symbol to the *Favourites* (check page 170) category.
- **Delete** – deletes the selected symbols from the ***Favourites*** category.

Pop-up menu– options menu for the selected symbol, which is available after right clicking on the mouse button on the selected symbol. Commands in this menu are similar to the tools menu.

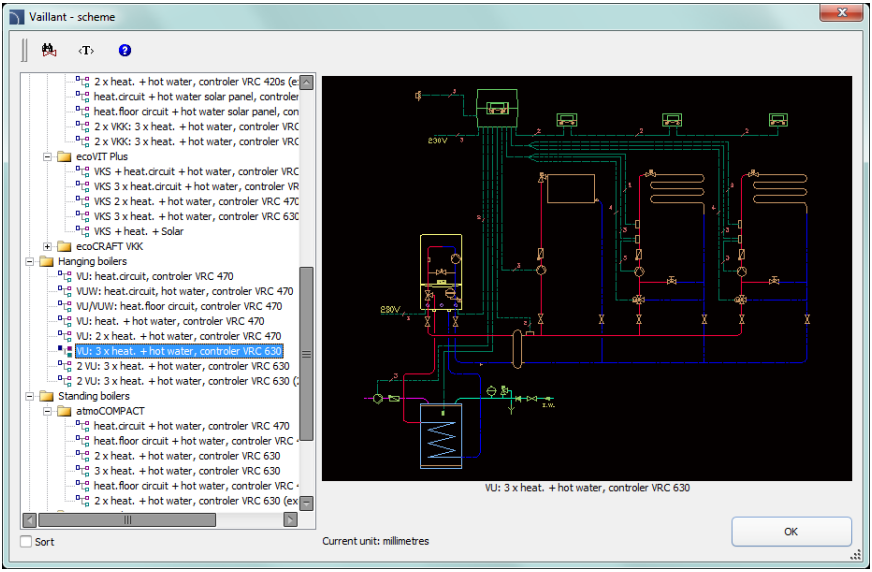


Angle/Rotation – possibility to specify a fixed angle when inserting symbols or enabling the rotation option, in which symbol's angle is defined by rotation during insertion.

Scale – specifies the size of inserted symbols. It is possible to enter the scale value or to indicate a symbol on the drawing whose scale user would like to use.

Schemes

Example of a scheme from the manufacturer's libraries:


Vaillant - scheme
2 x heat. + hot water, controler VRC 420s (e
heat.circuit + hot water solar panel, controler
heat.floor circuit + hot water solar panel, con
2 x VKK: 3 x heat. + hot water, controler VRC
2 x VKK: 3 x heat. + hot water, controler VRC
ecoVIT Plus
VKS + heat.circuit + hot water, controler VRC
VKS 3 x heat.circuit + hot water, controler VR
VKS 2 x heat. + hot water, controler VRC 470
VKS 3 x heat. + hot water, controler VRC 630
VKS + heat. + Solar
ecoCRAFT VKK
Hanging boilers
VU: heat.circuit, controler VRC 470
VUW: heat.circuit, hot water, controler VRC 470
VU/VUW: heat.floor circuit, controler VRC 470
VU: heat. + hot water, controler VRC 470
VU: 2 x heat. + hot water, controler VRC 470
VU: 3 x heat. + hot water, controler VRC 630
2 VU: 3 x heat. + hot water, controler VRC 630
2 VU: 3 x heat. + hot water, controler VRC 630
Standing boilers
atmoCOMPACT
heat.circuit + hot water, controler VRC 470
heat.floor circuit + hot water, controler VRC
2 x heat. + hot water, controler VRC 630
3 x heat. + hot water, controler VRC 630
heat.floor circuit + hot water, controler VRC
2 x heat. + hot water, controler VRC 630 (ex
Sort
VU: 3 x heat. + hot water, controler VRC 630
Current unit: millimetres
OK

In order to insert schemes from the manufacturers libraries run the **Manufacturers library** command from the main **CADprofi** toolbar or from the upper menu. Afterwards select the library of required manufacturer. In the program there are many already prepared boiler schemes prepared accordingly to ACV, Viessmann and Vaillant catalogs.

Marks

Marks and **Ventilation - Marks** commands are used to insert into the projects various marks that are used to on schematic drawings, views and 2D cross-sections.

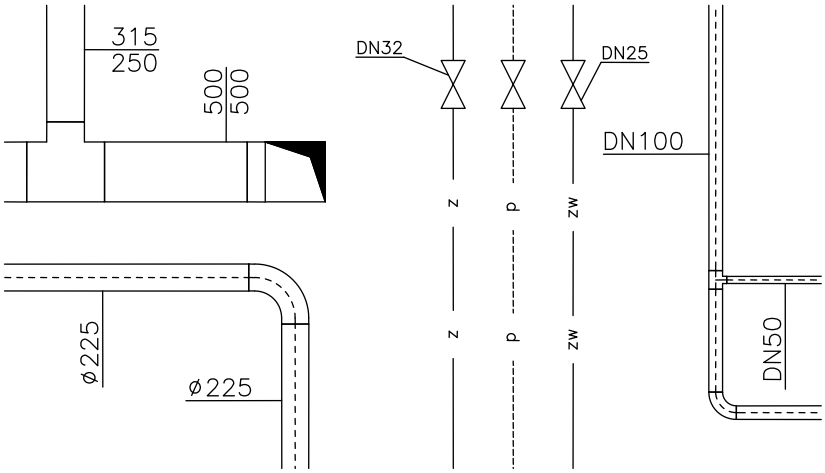
Automatic markings

CADprofi program possesses commands that are used for both object numbering and describing of objects. These commands allow to specify technical and identification parameters, which are later used when creating BOMs or when it's needed to obtain a variety of information about the project. Parameters that were added to the objects can be also used for automatic or semi-automatic description of drawings.

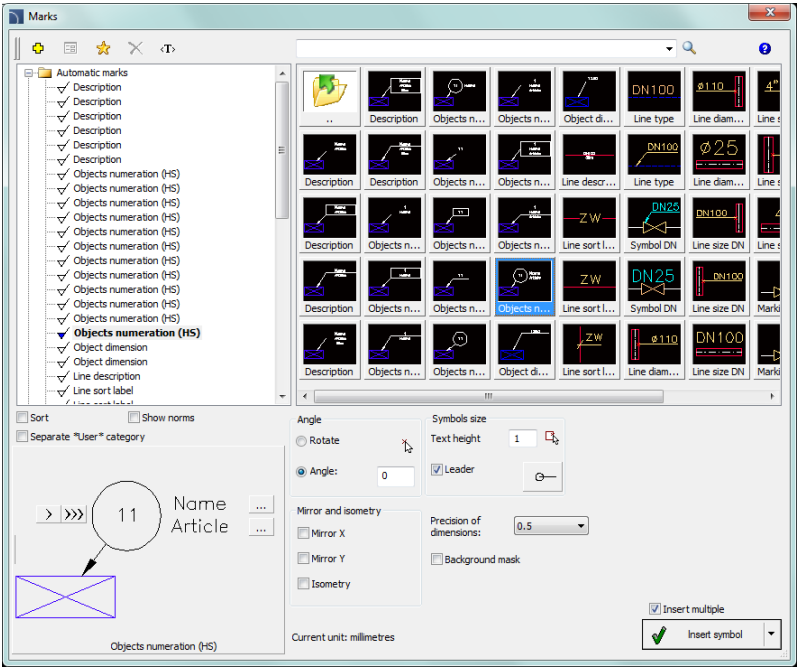
Symbols which gather information from objects and insert appropriate marks to the drawing are being used for this task. When inserting automatic marks user should indicate the described object and then specify the mark placing point.

For some symbols it is possible to enable the option that is used for inserting symbols with a leader. For describing various objects user should use appropriate symbols. For example the "description of rectangular duct cross section" symbol is being used to describe a duct cross-section size. When user uses this symbol for describing other objects, **CADprofi** program won't be able to read the needed data. In this particular case, the symbol would be described in an invalid way.

If it's necessary to mark multiple objects (such as ducts cross-section description) it is convenient to select the **Insert multiple** option.



Examples of automatic markings

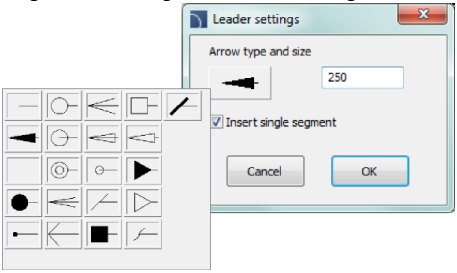


Marks dialog window

General functionality as well as symbols options were described in the **Symbols** (check page. 172) chapter. Additional options that are available in the **Marks** dialog window:

Leader – enables/disables usage of leader for descriptions. By clicking on the button, that symbolizes the currently set arrow type, user can change the following leader settings:

- **Arrow type**– symbol that is inserted in the leader starting point for example an arrow.
- **Arrow size** – determines the leader scale.
- **Insert single segment** –drawing of a one or multi-segment leader.



Text height – specifies the marks text height and scale. User can enter a value or indicate it from the drawing (indicate two points).

Precision of dimensions – defines the precision (decimal number) of all values of dimensions that are placed in marks. These dimensions are automatically retrieved from the indicated object.

Background mask – enables/disables the possibility to mask the symbol background by adding a WIPEOUT. This feature is helpful if user needs to highlight important information in the drawing.

Procedures

Inserting an automatic description (multivariant symbol)

1. Run the **Marks** or **Ventilation – Marks** command.
2. In the **Marks** dialog window select **Marks** category from the list of available categories. In the top-right corner the category content will be displayed as thumbnails.
3. Click on a thumbnail of a chosen symbol, in the bottom-left corner you will be able to see the symbol preview.
4. (Optional) Change the data type by clicking the **Browse** button to select the data from the list of available data in the **Select** dialog window.
5. Specify the parameters: **Angle**, **Mirror**, **Isometry** and **Scale**.
6. (Optional) Enable and change **Leader** settings.
7. Enable/disable the **background mask**.
8. Enable/disable the **Insert multiple** option.
9. Click the **Insert symbol** button, to insert the desired mark into the drawing.
10. Indicate the symbol/object whose data should be inserted into the drawing.
11. Indicate the mark insertion point.
12. (Optional) If the **Insert single segment** option (in the leader settings) is turned on then indicate a series of points that are segments ends or click the right mouse button to continue.
13. (Optional) Specify the mark rotation angle.
14. Mark will be inserted in accordance to the selected settings.
15. (Optional) If in the dialog window the **Insert multiple** option is enabled then repeat the steps 10-13 in order to insert an additional mark or end the command by clicking the **Esc** key.

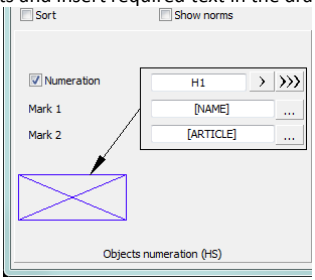
Marks

To edit marks user can use **CADprofi** commands such as **Edit symbols**, **Attributes and descriptions** (check page 34) or CAD program commands such as Edit attributes - _DDATTE.

Multivariant marks and object numbering

Many mark symbols are multivariant symbols, for which user has got a possibility to customize the appearance and to select additional options. For these symbols in the dialog window there are selection fields and text fields buttons available that allow to specify the variant or adoption of symbol to current needs.

An example of such multivariant symbol is the "Objects numeration" symbol. This symbol can number objects and also retrieve from them required parameters (e.g. name, type etc.). In this way it is possible to quickly number many objects and insert required text in the drawing.



CADprofi program automatically detects numbered objects in the drawing, thanks to which in the dialog window the first available number › from the specific branch (e.g. HS - heating and sanitary installations) is being displayed. The ››› button allows to select the next highest number.

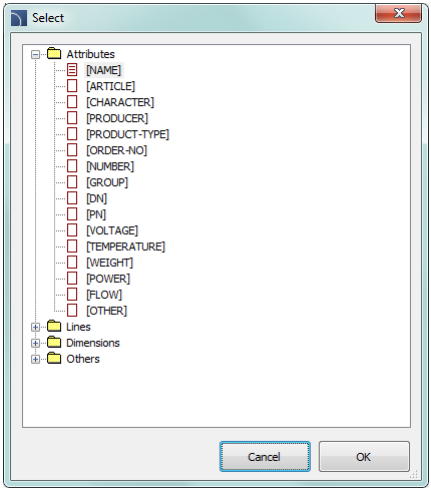
By adding different number prefixes, user can create many numbered sequences (e.g. one Z1, Z2, Z3 sequence for valves and one **P1**, **P2**, **P3** for pumps).

Inserting symbols with the **Insert mupltiple** option will cause the symbols to have numbers added to them with increased numeration (e.g. from **1** to **10**).

The **Mark 1**, **Mark 2** fields allow to add in the selected symbol any text or information retrieved from the indicated object. Selecting the information type will be possible after pressing the ... button that is located next to the field.

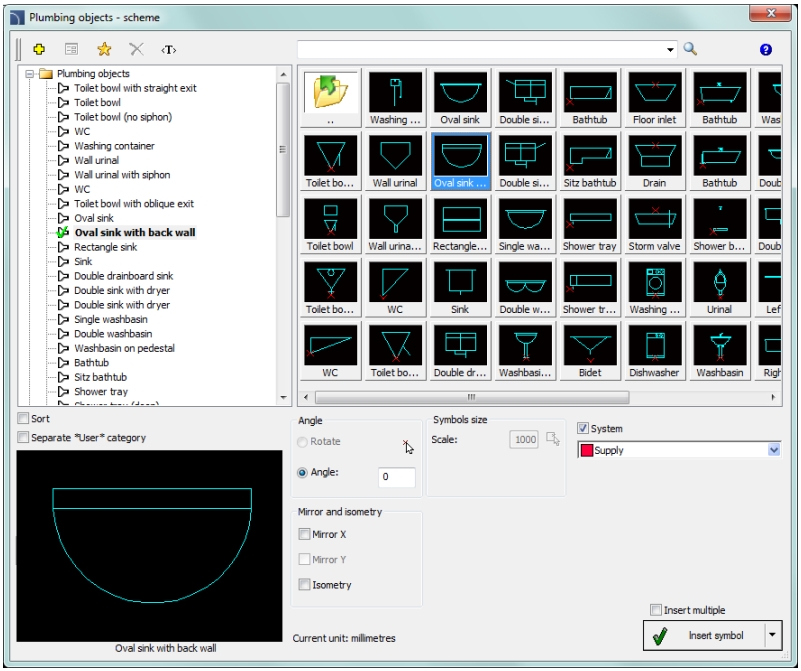
In the newly opened window user can select the following information:

- **Attributes** – indicated object attributes values.
- **Lines** – line type or installation kind.
- **Dimensions** – dimensions retrieved from the indicated object.
- **Others** – for example the CAD program system variable values.



Plumbing objects – scheme

Special kind of symbols are plumping objects that are used in plumbing strand diagrams.
In strand diagram drawings it is necessary to maintain the vertical scale on the printouts.
Therefore, the sanitary symbols are being inserted in standard size. This size depends on the used design unit (mm, cm, m) that's why the scale change option is blocked.
Plumbing objects can be inserted next to the risers from the right or left side. For some symbols it is necessary to turn on the **Mirror X** option. Symbols of plumbing objects should be inserted with "0" angle.



Plumbing objects – scheme dialog window

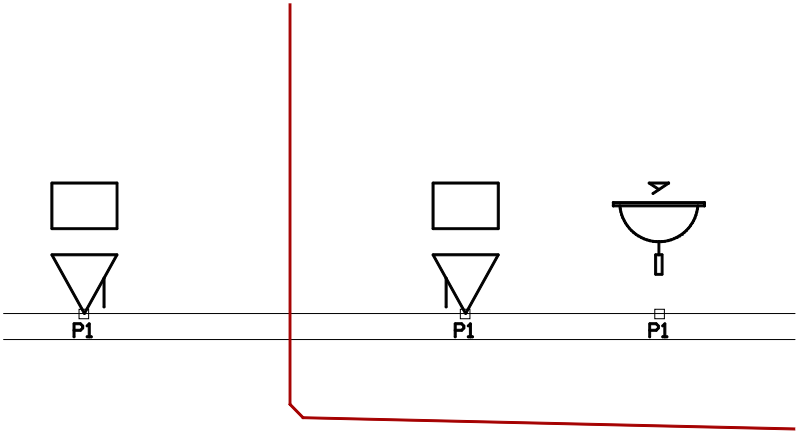
The basic functionality and symbols options are being described in the **Symbols** (check page 172) chapter.

Procedures

Inserting symbol

1. Run the **Plumbing ojects - scheme** command.
2. In the **Plumbing ojects - scheme** dialog window select the needed symbol from the appropriate category.
3. Click the thumbnail of the symbol that you would like to insert into the drawing. In the down-left corner you may see the preview of selected symbol.
4. Select the appropriate **system**.
5. (Optional) Enable the **Mirror X** option.
6. Click the **Insert symbol** button, to specify the object insertion point (**P1**).
7. Symbol will be inserted into the drawing.

Plumbing objects – scheme



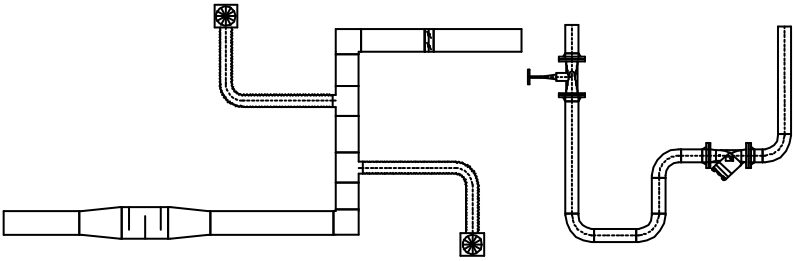
Notice

For all plumbing objects the insertion point is defined at the floor level. For this reason, before placing the first symbol user should draw a ceiling cross-section.

During inserting plumbing objects symbols on strand diagrams user should always indicate the point that lies upper floor edge (P1).

2D views of piping and ventilation installations

Creating drawings of 2D installations is related to drawing lines, adding objects and descriptions. **CADprofi** program enables automatic drawing of line strings and fittings of any kind in 2D view. It also allows user to quickly connect lines and to edit them. It is also possible to insert armature views in previously drawn lines with an ability to automatically insert reduction. All objects are inserted on appropriate layers that define their line kinds or system (supply, return, air etc.).



An example of 2D view

CADprofi program possesses different objects:

- **Radiators** – it is possible to use radiators on 2D projections and strand diagrams.
- **Plumbing objects**– objects used in designing water and sewage installations.
- **Parametrical objects**– these objects represent actual views of devices such as boilers, tanks, block pumps etc. For parametrical objects it is possible to specify their technical data and all dimensions.
- **Parametrical armature** just like with the objects, parametrical armature represents actual devices view (armature, lines equipment, flanges, in-line 2D lines). The main feature of these objects is the ability to insert them in 2D lines.
- **Objects from manuafcturer's libraries** – they are available in the **Products library** command. Products libraries are created together with manufacturers and therefore contain a very comprehensive source of objects (blocks) that are used in designing. Products libraries are constantly being updated and are made available by manufacturers on the internet. After installing **CADprofi** program it is possible to install additional libraries at a later time that will be added to the program.
- **Objects defined by users** – every user can add to the program any objects used in designing.

2D lines and fittings

In **CADprofi** program it is possible to create 2D views and cross-sections of pipes and ducts. When drawing installations lines and fittings (elbows, reducers, tee-pieces) are available. Each fitting is a parametrical object, which is being drawn with appropriate program procedures. Thank to this it is possible to draw fittings with specified dimensions. For pipes and fittings databases are available that contain definitions of standardized elements of chosen manufacturers.
Lines and fittings are available in the following commands:

- **2D Pipes - plan** – this command contains typical pipes and fittings used in designing piping installations.
- **2D Ducts** – this command contains ducts and fittings of circular and rectangular shape used in designing ventilation and air conditioning installations.
- **2D Pipes/Ducts sequence** – this command allows to draw many lines segments with automatic insertion of elbows.
- **2D Join pipes/ducts** – this command is used for a convenient connecting of lines with an automatic insertion of an appropriate fittings.

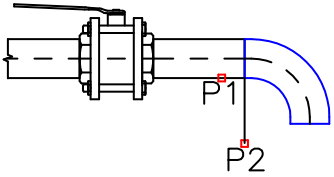
In all these command, a similar dialog window is being used. This dialog window has got a list that is used to define parameters for fittings. A set of available parameters depends on the type of selected fitting. The meaning of each individual parameter is similar for many fittings, that's why we ask users to read carefully all the explanations explained in this chapter so it will be possible to fully exploit the program opportunities.

Inserting lines and fittings

Both **2D Pipes - plan** and **2D Ducts** commands allow to insert single fittings or lines to the drawing.
After running the command it is possible to select the needed fitting or line and to specify the dimensions and insertion options. **CADprofi** program also possesses features that automate connecting elements to each other. It is also possible to automatically adjust the line size and the system kind when connecting lines to other elements. The below examples were created with using these functions.

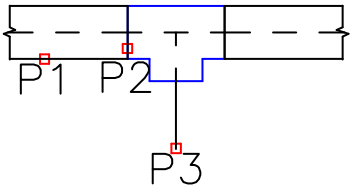
Automatic connection of a fitting to an installation element

1. Indicate an element to which you would like to connect a fitting (**P1**).
2. Specify the fitting insertion direction (**P2**).



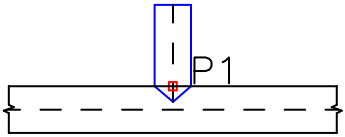
Inserting a tee-piece with the „Insert in line" option

1. Indicate in line in which you would like to insert a tee-piece (**P1**).
2. Specify the fitting insertion point (**P2**).
3. Specify the tee-piece direction (ramification) (**P3**).



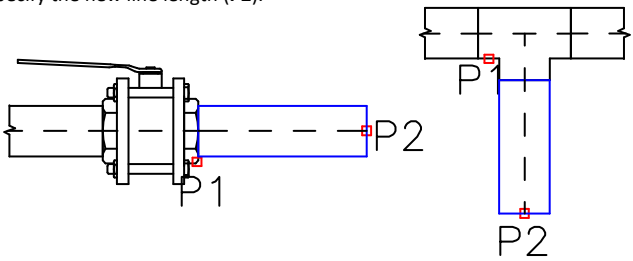
Inserting a coupling into an existing line

1. Specify the insertion point (**P1**).
2. Automatic adjustment of the angle that was specified in the dialog window (**alfa**).



Inserting a line with the „Join object" option enabled

1. Indicate an object to which you would like to connect a line (**P1**).
2. Specify the new line length (**P2**).

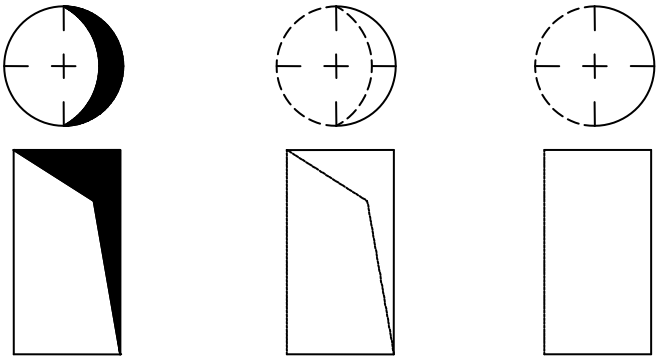


When connecting pipes or ducts to other installation elements, both **Diameter fit** and **Width and height fit** options are useful. Thanks to them the inserted line will always have the same size as the indicated object.

Z+/Z– lines and fittings

Z+/Z- lines show vertical line segments (along the Z axis). For these lines on projections only the appropriate cross-section symbol is visible. For this reason, the Z+/Z- line length should be specified in the dialog window before inserting lines to the drawing.

For Z+/Z- lines and fittings it is possible to additionally specify the symbol cross-section (Z+, Z- or without symbol) and the insertion point (beginning, axis, end).



Z+ *Z–* *Without symbol*

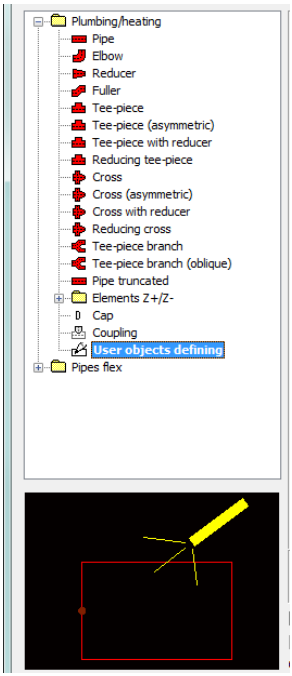
Copying fittings dimensions

During drawing it is often necessary to insert elements similar to other elements that are already inserted in the drawing. In this case, it is convenient to copy dimensions from other fittings. After pressing the **Copy dimensions** button, user should indicate a desired fitting. It is important to avoid identifying fittings that have completely different parameters than the currently selected element in the dialog window. In such cases, user may experience an ambiguity in the data interpretation.

For example: if in the dialog window user has selected an elbow, and when copying dimensions user will indicate a reducer, then **CADprofi** won't be able to determine the curvature radius and the elbow angle, because reducer doesn't posses such parameters.

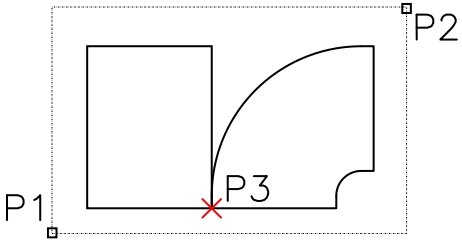
Defining user own fittings

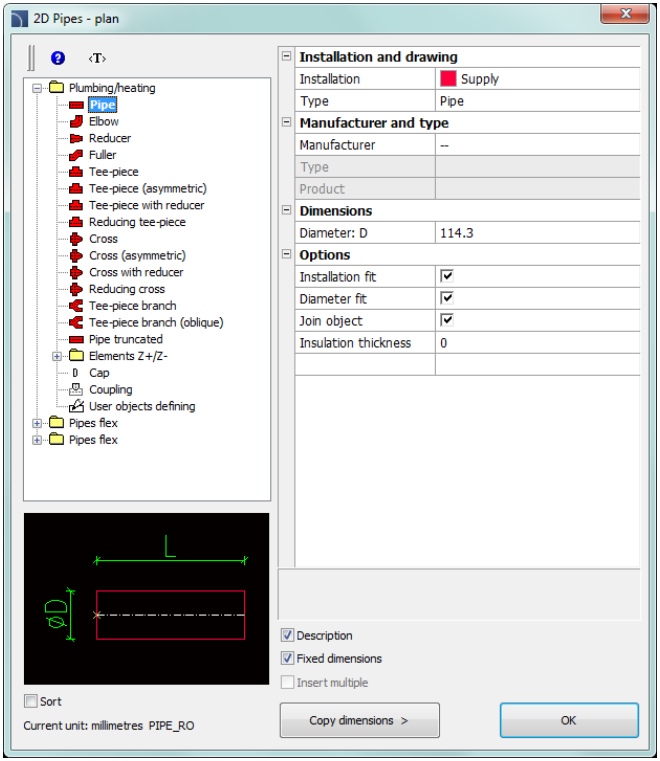
When designing installations it is sometimes needed to use non-standard fittings. In this case, user can define his own fittings, which will be taken into account during numbering or during creating BOM.
In order to define a new additional fitting, user should draw it first and then run the **2D Pipes - plan** or the **2D Ducts** command. After that user should choose the User object defining option.
For the newly created fitting user should choose a name. Optionally user can also specify the manufacturer, type and product parameters.
After clicking the OK button, user should select objects from the list that the new fitting is being made up from and to choose the base point.
If a fitting is often used then it is possible to add it to the program library by using the **User block wizard** (check page 87).



An example of defining a tee-piece (ramification).

Defined fitting consists of a rectangular segment and an arch. In order to draw a new fitting it is convenient to use the **2D Ducts** command, thanks to which it is possible to insert a rectangular duct and a right arc with a 90° angle.
In order to use a new fitting in installations while maintaining colors and other characteristics all elements should be moved to the "0" layer. It is necessary to have the fitting color the same as the installation color (e.g. air, exhaust air). After that user should run the **2D Ducts** command and select the **User object defining** option. In the **Name** field enter a name for example "rectangular tee".
After pressing the **OK** button, select the elements that belong to the defined fitting (**P1**–**P2**) and specify the fitting base point (**P3**).
When the fitting is being defined it is also automatically inserted on a layer that belongs to the selected installation.





2D Piping – plan/2D Ducts dialog window

2D Piping – plan/2D Ducts contains the following items:

Categories tree – a list of all available elements divided into categories.

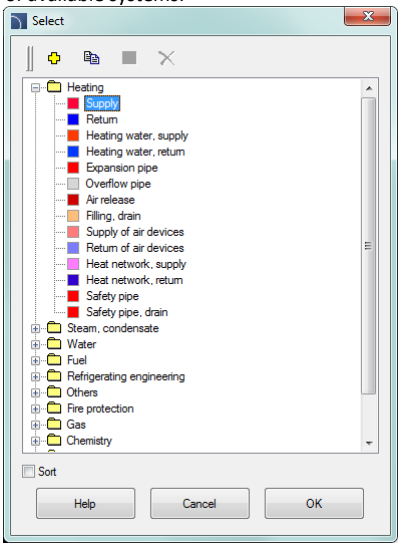
Preview – displays the preview of the currently selected element. Right-clicking on the preview window will zoom the view.

Sort – enables/disables alphabetical order of categories placement and their content.

Fixed dimensions – enables/disables the preserve of a basic dimension after inserting objects such reducers. After inserting such element in which the line dimension is being reduced in the next time user runs the command the application will automatically change the default diameter into a reduced diameter (**D2**). This process is very convenient when designing an installation sequence. However, if user wants to insert a fitting that has an unchanged main cross-section then enabling the **Fixed dimensions** option will prevent an automatic change of cross-section.

Installation and how to draw:

- **Installation** – possibility to choose the installation kind or define a new one. Initially on the drop-down list only the most important **CADprofi** systems will be included. However, user has the possibility to develop the list with his own elements (by using the **Add to most recently used** option) in the **Edit line data** (check page 158). dialog window. Selecting the **Others/Edit** option will open a new dialog window with full list of available systems.



Define new – allows to add a new position (line) to the list as a user line. Afterwards it opens a new **edit** window and allows user to edit the line data.

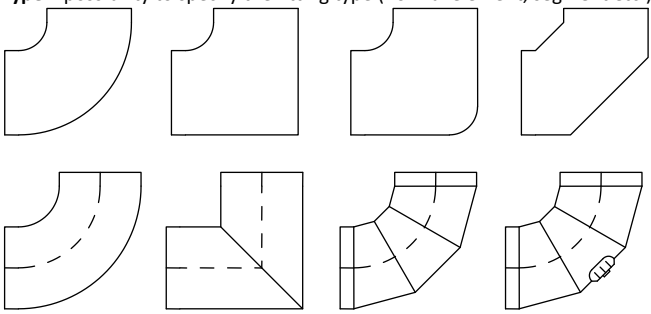
Copy – copies the selected line with all its parameters, therefore creating an element made by user.

Edit – opens the edit properties window of user created lines.

Delete – deletes the selected line from the list.

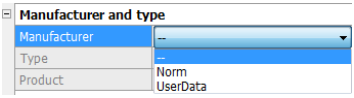
List of available lines (installations) is similar to the list of lines available in the **Pipes/Ducts – scheme** (check page 148) commands. Any changes made in one of these commands will also appear in others.

- **Type** – possibility to specify the fitting type (normal element, segment etc.).

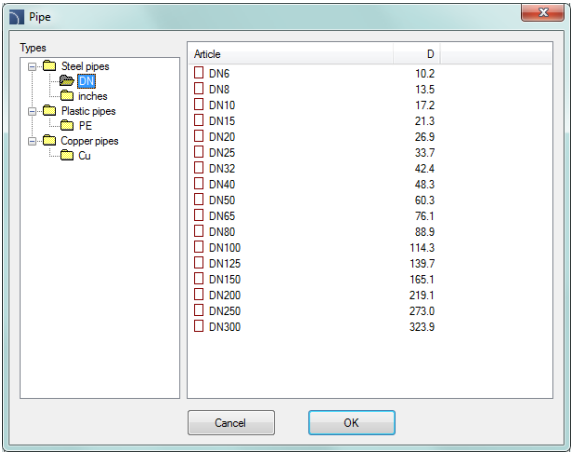


Elbow types available in the program

- **Manufacturer and type** – selection of items from manufacturers standards or databases. **CADprofi** program allows to draw fittings and 2D lines, whose dimensions can be specified in two ways:
  1. Drawing based on dimensions specified by user.
     To enable this mode, in the **Manufacturer** field select „– –".
  2. Drawing based on manufacturers' standards or databases. In order to activate this mode select the required manufacturer (or standard). After selecting the manufacturer it is not possible to enter you own dimensions (this option is blocked).



     After selecting a specific manufacturer or standard a dialog window will appear with available fittings or lines.



Element selection dialog window

Dimensions – In **CADprofi** program it is possible to draw fittings with any dimensions. To do this, in the **Manufacturer** field select „– –"
When drawing is based on user selected dimensions then two field in which user can enter values are available. button allows to select typical values, for example standardized piping diameters or ducts cross-sections. The straight lines length is determined in the drawing by indicating two points.

Options – additional options that automates the drawing process:

- **Join object** – automatic connection of inserted element to existing objects (line, fittings or armature). When connecting, **CADprofi** program will automatically determine the rotation angle and the element insertion point.
- **Installation fit** – automatic determination of installation kind based on object, to which the inserted element is being connected. Installation kind set in the dialog window is being ignored.
- **Diameter fit** – automatic identification of installation diameter based on object to which a line is being connected. In the connection point it is possible to insert a reducer or to ignore the line diameter from the dialog window.
- **Insert in a sequence** – gives the possibility to insert fittings into an existing line or into their extension. If this option is disabled then fittings are inserted on the end of the line.

Thickness of insulation – determines the insulation thickness of inserted fitting or line. The default value **0** disables insulation.

Description – enables/disables the display of description of the currently selected option in the dialog window.

Copy dimensions – copies dimensions from a line/fitting indicated in the drawing.

OK – closes the window and allows to insert an element to the drawing.

Procedures

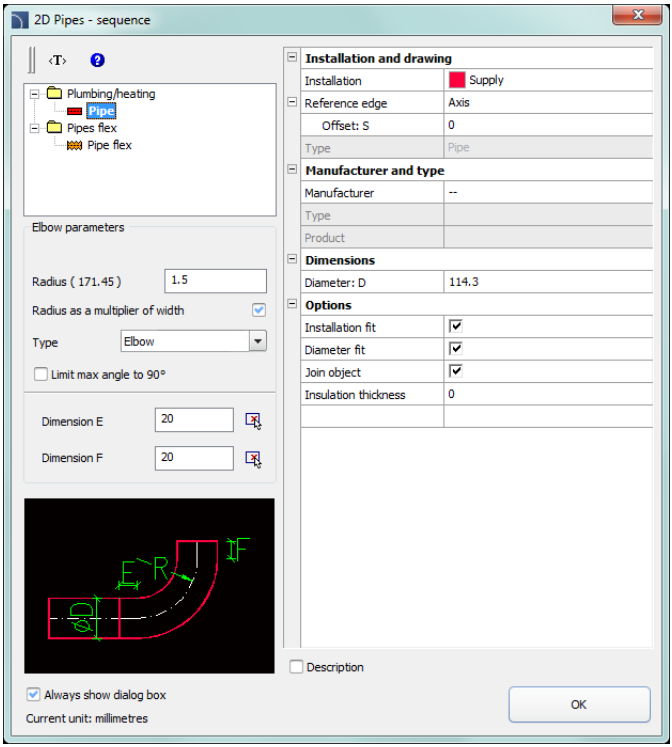
Inserting lines

1. Run the **2D Pipes - plan** or **2D Ducts** command.
2. In the **2D Pipes - plan/2D Ducts** from the list of available categories select the **Pipe** from the **Piping/heating** category or **Pipe flex** from the **Pipe flex** category.
3. From the **list of installations** select the needed kind of installation in order to specify the color and layer for a new line.
4. (Optional) Specify the values for the **Manufacturer and type** items.
5. In the **Dimensions** field specify the **Diameter** value for the selected line.
6. (Optional) Click the **Copy dimensions** button and indicate a line on the drawing from which you would like to copy the diameter value.
7. (Optional) Enable/disable the **Installation fit**, **Diameter fit**, and **Join object.**
8. (Optional) Specify the **Thickness of insulation**.
9. Click the **OK** button to start drawing a line.

2D lines and fittings

Drawing line sequences

The **2D Pipes - sequence** and **2D Ducts - sequence** commands allow to create piping sequences with automatic insertion of elbows that have parameters defined by user. These commands have the ability to disable the display of the dialog box, allowing user to immediately start drawing lines sequences with the last used settings or with settings that matches the objects to which they will be attached (with the **Installation fit** and **Diameter fit** options enabled).



2D Pipes – sequence/2D Ducts – sequence dialog window

Graphical layout and basic functionality of the dialog window is similar to the **2D Piping – plan/2D Ducts** (check page 194). Additional options available in this command:

Elbow parameters – a group of options that allow to specify parameters of elbows drawn in 2D.

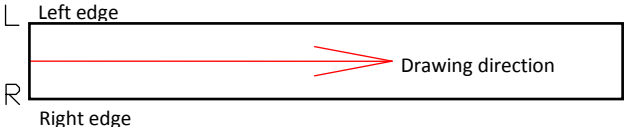
- **Radius** – allows user to determine the radius or radius ratio for elbows inserted between each line segments.

- **Radius as a multiplier of width** – enables/disables the possibility to determine the radius through the width multiplier.
- **Type** – allows user to choose the elbow type that will be used in the line sequence.
- **Limit max angle to 90°** – enables/disables the possibility to reduce the line sequence elbow angle to 90°.

Dimension E/Dimension F – allows to specify both E and F dimensions (that are visible on the preview). button allows to specify the dimension by indicating two objects on the drawing.

> ***Hint***
>
> ***If the „Always show dialog box" option was disabled then if user wants to enable it again he has to choose Settings (abbreviation „S" in the command line) after running the command.***

Reference edge – right or left edge. Edge is indicated accordingly to the drawing direction.



Procedures

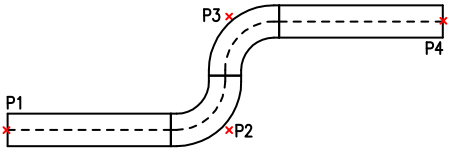
Inserting lines

1. Run the **2D Pipes - sequence** or **2D Ducts - sequence** .
2. (Optional) If the dialog window didn't appear then start drawing or select the **Settings** (S key) option, to change the settings.
3. In the dialog window choose the needed kind of line.
4. Choose installation kind.
5. (Optional) Specify the values for **Manufacturer and type** controls.
6. Specify the line **Diameter** (or cross-section).
7. (Optional) Enable/disable the **Installation fit**, **Diameter fit**, and **Join object** functions.
8. (Optional) Specify the **Thickness of insulation**.
9. Click the **OK** button to start drawing a line sequence by indicating the points.

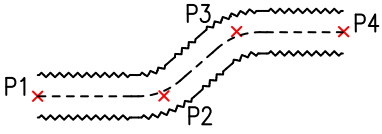
Drawing line sequences

How to draw

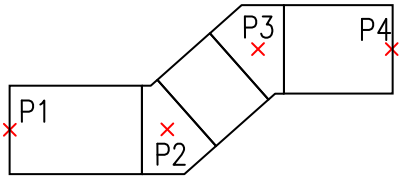
When drawing users should indicate points (**P1**, **P2** etc.). If you draw by the axis and without an offset line segments are being inserted along with the indicated points.



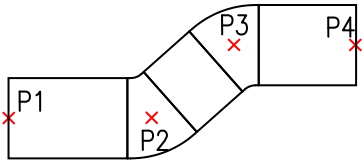
Pipe



Pipe flex



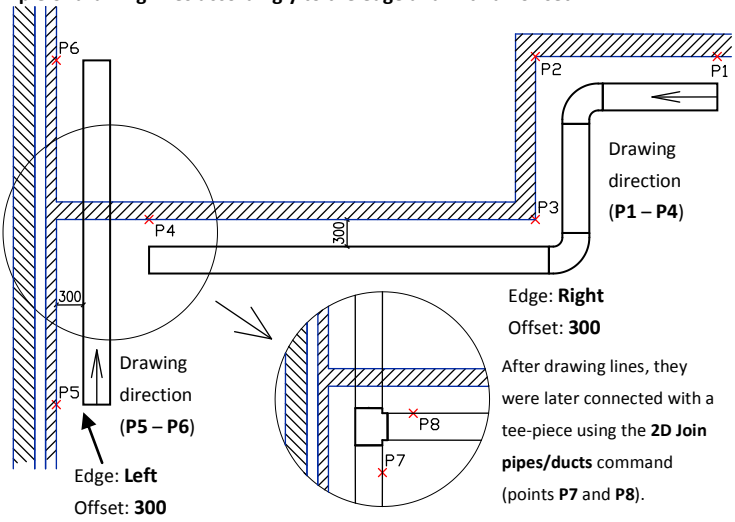
Rectangular pipes with elbows



Rectangular pipes with arches

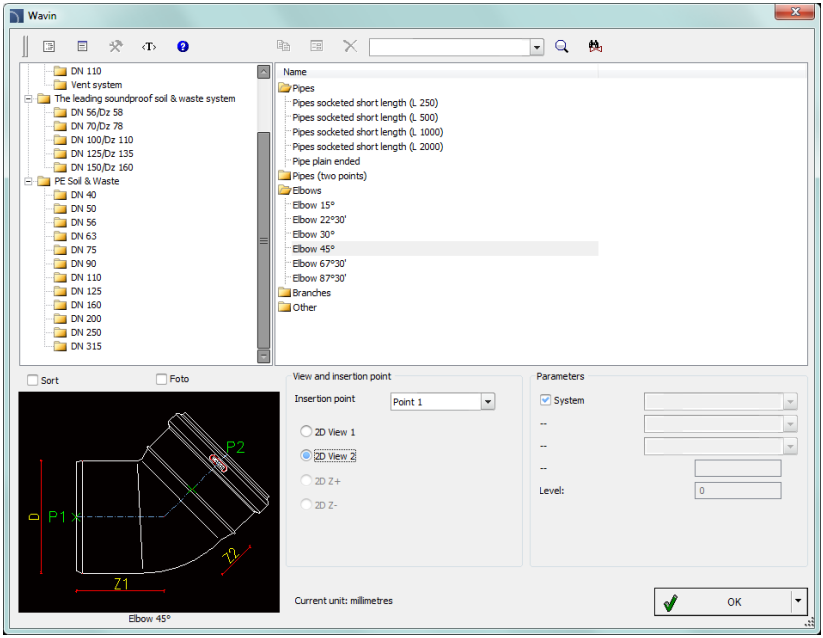
When drawing with an offset the line sequence is being drawn with a specified distance from the indicated points.

An example of drawing lines accordingly to the edge and with an offset:



Drawing sewerage installations views

Sewerage installations can be drawn with the command used for pipes drawing. In this command it will be needed to "manually" specify the dimensions for pipes and fittings.
View of such installations will be only an approximation of a real installation.
It is also possible to use the Wavin library, which is available in the **Manufacturers library** command.

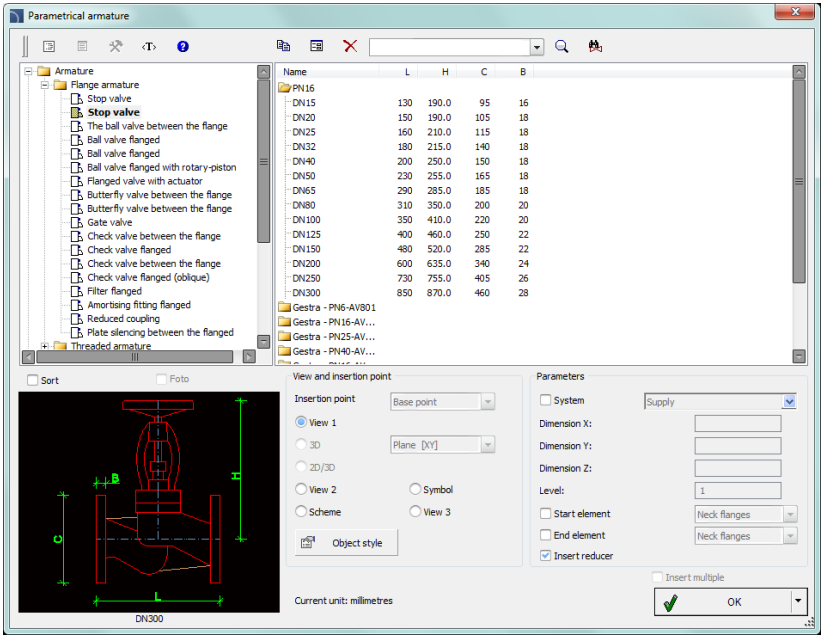


Wavinlibrary dialog window

PVC, AS and welded HD-PE elements can be found in the library.
All elements have appropriate points that allow user to automatically connect elements to other elements that were already inserted in the drawing. What is more, program validates if the connection is correct and alerts if there are problems or elements that don't match with each other.

Parametrical armature

The **Parametrical armature** command allows to insert to the drawing armature views, "in-line" pumps and other piping objects. For each elements, databases are available that contain standardized elements and elements based on manufacturers types. All elements are parametrical objects. It means that when user inserts an object into the drawing he has the possibility to determine dimensions such as length, width, connection sizes or the dimensions of a manual valve knob as well as other dimensions. This feature gives user a lot of opportunities, because through self-determination of dimensions, user can use any devices, even those that are not available in standard program databases.



Parametrical armature dialog window

Parametrical armature dialog window contains the following elements:

List of categories – displays a list of available elements grouped in categories and subcategories.

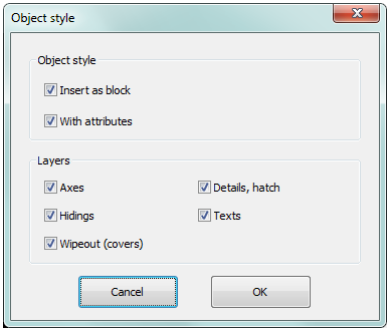
Sort – enables/disables categories alphabetical order.

Detailed list – list of elements available for the selected category.

Preview – displays the preview of the currently selected element. Right-clicking on the preview window will zoom the view.

View and insertion point – view selection and object insertion points. Available options depend on the type of object.

Object style – options that allow to specify the object style and the level of object detail.



Object style dialog window

- **Insert as block** –allows user to insert an element that is exploded into component parts or a one single object (block).
- **With attributes** – enables/disables adding of attributes to the inserted object.
- **Axes** – enables/disables an option to draw axis in the inserted object.
- **Hidings** – enables/disables drawing elements that are hidden in an object.
- **Wipeout (covers)** – enables/disables addition to the inserted elements **Covers** (WIPEOUT) type objects.
- **Details, hatch** – enables/disables drawing of hatches and object details such as holes in the flanges, arrows that indicate the direction flow and others.
- **Texts**– enables/disables drawing of potential texts in objects.

> ***Notice***
>
> ***All „Object style" settings will be remembered and used when user runs the command again.***

Parameters – options that specify additional parameters of inserted object.

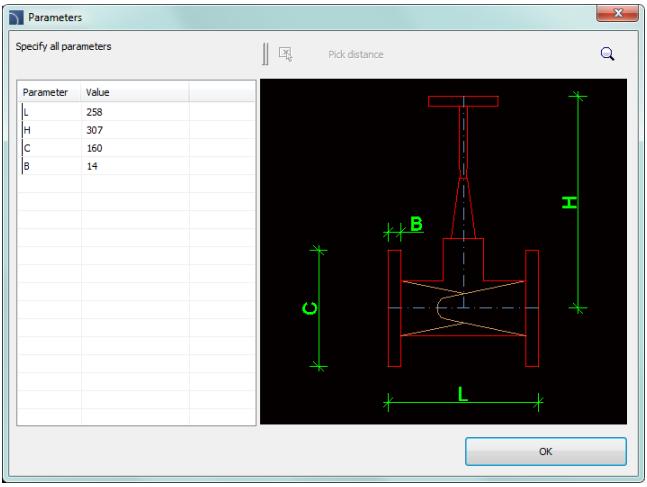
- **System** – if in the program options, the **extended layer name structure** (check page. 29) is enabled then objects will be inserted on layers depending on the kind of installation (system). In case of inserting objects into an existing line then the **System** option is being ignored and the object layers depends on the line kind.
- **Start/End element** –gives the possibility to connect to the object additional elements. For example this option allows to insert together the valve flange together with properly selected flanges (at the start and end of the valve).

- **Insert reducer** – automatic insertion of a reducer. This option is used when inserting and object into an existing line if the object nominal diameter is different from the line diameter.

OK – allows user to insert selected object into the drawing.

Insertion options – expands the list of available insertion options:

- **Insert (specify style for object)** – inserts selected object with a possibility to change options in the **Object style** dialog window.
- **Insert (specify all dimensions)** – inserts selected object with possibility to specify all dimensions that can be edited. It is also possible to save the user dimensions by Creating and extending the user type of series **Creating and extending user type of series** (check page 88).



Parameters dialog window

When user selects this option, a dialog window will appear allowing to enter values for each dimension. button allows to specify the selected dimension directly on the drawing (by indicating two points).

Procedures

Inserting armature

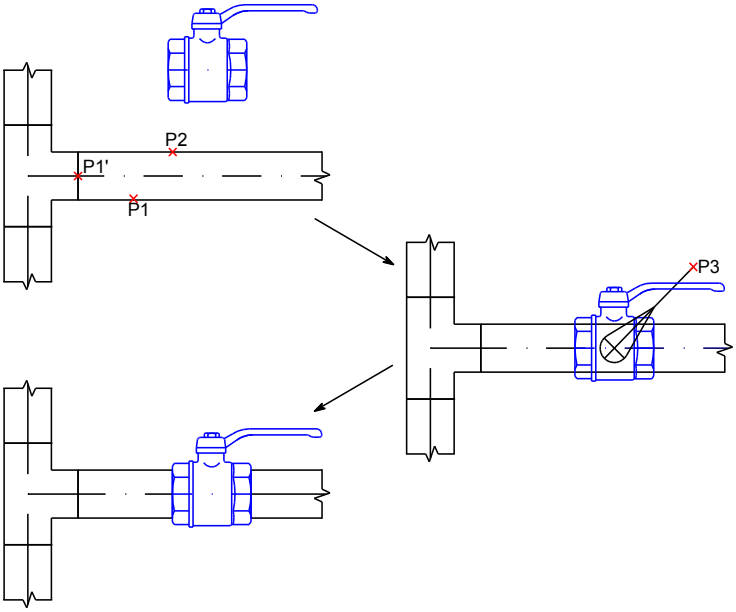
1. Run the **Parametrical armature** command.
2. In the **Parametrical armature** dialog window select the needed object category. After that the element preview will be displayed and on the detailed list you will be able to see all available element types and sizes.
3. (Optional) If the list of available types possesses subcategories double click on a category will expand it and you will be able to see its content.
4. (Optional) Right click on any item and from the menu choose the **Expand all/Collapse all** option.
5. Choose the needed element form the detailed list.
6. (Optional) Change the **View** for the selected element.
7. (Optional) Change the settings for the **Object style** options.
8. (Optional) Switch on the **System** option and select the installation from the list.
9. (Optional) Set the: **Start element, End element** and **Insert reducer**.
10. Click the **OK** button to insert selected object into the drawing.
11. (Optional) Expand the button and select **Insert (specify all dimensions)** to specify dimensions of inserted object.
12. (Optional) Expand the button and select **Insert (specify style for object)** to prior before inserting an object make changes in the **Object style** option.
13. Click on the edge of a line, in which you want to insert a selected element, and then specify the object insertion point in the indicated line.
14. (Optional) Click at any point in the drawing to insert the element.
15. Specify the element rotation angle or the insertion direction if the element is inserted into a line.

Parametrical armature

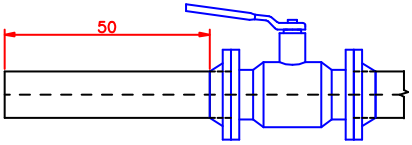
Inserting armature in a 2D line

In order to insert armature into a line user should:

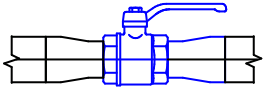
1. Indicate (**P1**) point. Application will automatically choose the nearest line end, as the reference point (**P1'**).
2. Afterwards specify the actual insertion point. It is possible to enter in the command line the distance from the reference point (from the beginning of the line) or to indicate a point (**P2**). If the point will be indicated further than the line length then program will automatically extend the line and insert the armature at its end.
3. For many objects it is also possible to indicate the (**P3**) point in order to determine the direction of inserted object.



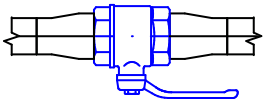
An example of inserting armature into lines:



Threaded valve with a nominal diameter corresponding the line diameter.



Threaded valve with a nominal diameter smaller than the line diameter with the **Insert reducer** option enabled (automatic reducer insertion).

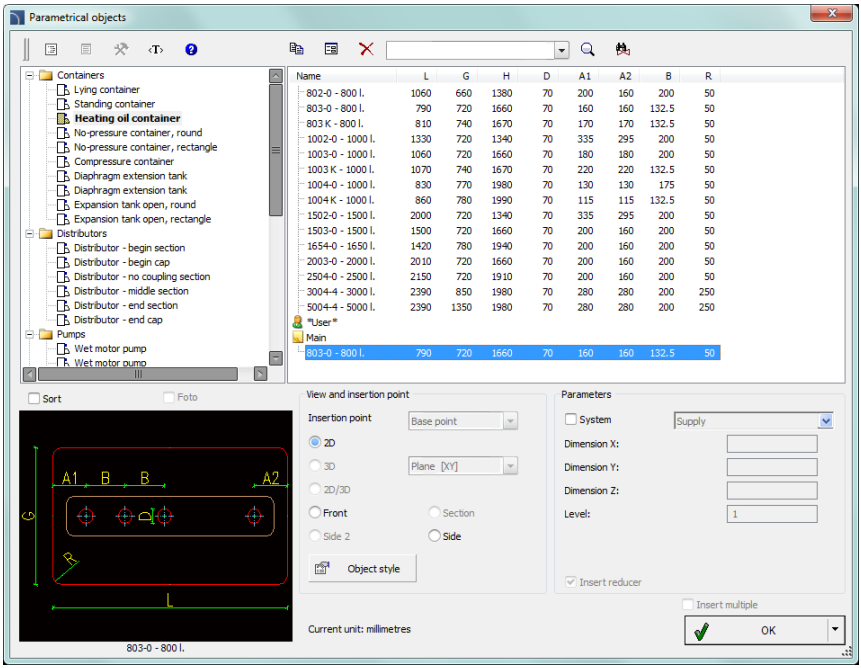


Threaded valve with a nominal diameter bigger than the line diameter with the **Insert reducer** option enabled (automatic reducer insertion).

Parametrical objects

The **Parametrical objects** command is used for inserting various objects such as tanks, valves, pumps, fans, boilers, water heaters etc. to the project. These objects are mostly not inserted into lines.

Just as with the **Parametrical armature** command installation objects are parametrical. It means that during insertion user has the possibility to specify all the dimensions or to choose a specific type from the database.


Parametrical objects
Containers
Lying container
Standing container
Heating oil container
No-pressure container, round
No-pressure container, rectangle
Compressure container
Diaphragm extension tank
Diaphragm extension tank
Expansion tank open, round
Expansion tank open, rectangle
Distributors
Distributor - begin section
Distributor - begin cap
Distributor - no coupling section
Distributor - middle section
Distributor - end section
Distributor - end cap
Pumps
Wet motor pump
Wet motor pump
Name L G H D A1 A2 B R
802-0 - 800 l. 1060 660 1380 70 200 160 200 50
803-0 - 800 l. 790 720 1660 70 160 160 132.5 50
803 K - 800 l. 810 740 1670 70 170 170 132.5 50
1002-0 - 1000 l. 1330 720 1340 70 335 295 200 50
1003-0 - 1000 l. 1060 720 1660 70 180 180 200 50
1003 K - 1000 l. 1070 740 1670 70 220 220 132.5 50
1004-0 - 1000 l. 830 770 1980 70 130 130 175 50
1004 K - 1000 l. 860 780 1990 70 115 115 132.5 50
1502-0 - 1500 l. 2000 720 1340 70 335 295 200 50
1503-0 - 1500 l. 1500 720 1660 70 200 160 200 50
1654-0 - 1650 l. 1420 780 1940 70 200 160 200 50
2003-0 - 2000 l. 2010 720 1660 70 200 160 200 50
2504-0 - 2500 l. 2150 720 1910 70 200 160 200 50
3004-4 - 3000 l. 2390 850 1980 70 280 280 200 250
5004-4 - 5000 l. 2390 1350 1980 70 280 280 200 250
User
Main
803-0 - 800 l. 790 720 1660 70 160 160 132.5 50
Sort
Foto
A1 B B A2
G
R
L
803-0 - 800 l.
View and insertion point
Insertion point
Base point
2D
3D
Plane [XY]
2D/3D
Front
Section
Side 2
Side
Object style
Parameters
System
Supply
Dimension X:
Dimension Y:
Dimension Z:
Level:
1
Insert reducer
Insert multiple
Current unit: millimetres
OK

Parametrical objects dialog window

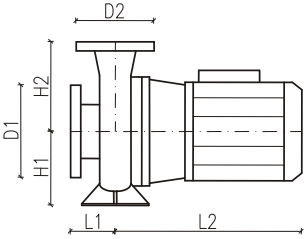
Graphical layout and the base dialog window functionality is the same as in the **Parametrical armature** (check page 202) dialog window.

Procedures

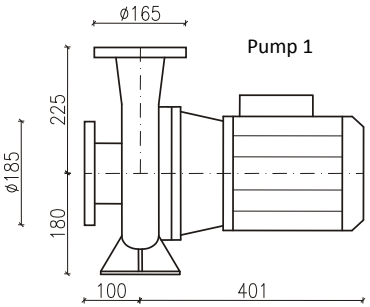
Inserting parametrical objects

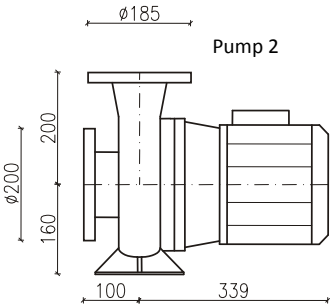
1. Run the **Parametrical objects** command.
2. In the dialog window select the needed object category. After that the element preview will be displayed and on the detailed list you will be able to see all available element types and sizes.
3. (Optional) If the list of available types possesses subcategories double click on a category will expand it and you will be able to see its content.
4. (Optional) Right click on any item and from the menu choose the **Expand all/Collapse all** option.
5. Choose the needed element form the detailed list.
6. (Optional) Change the **View** for the selected element.
7. (Optional) Change the settings for the **Object style** options.
8. (Optional) Switch on the **System** option and select the installation from the list.
9. Click the **OK** button to insert selected object into the drawing.
10. Specify the insertion point.
11. Specify the object rotation angle.

Examples of parametrical objects inserted with user own dimensions



D1	ø185 (DN65)	ø200 (DN80)
D2	ø165 (DN50)	ø185 (DN65)
H1	180	160
H2	225	200
L1	100	100
L2	401	339



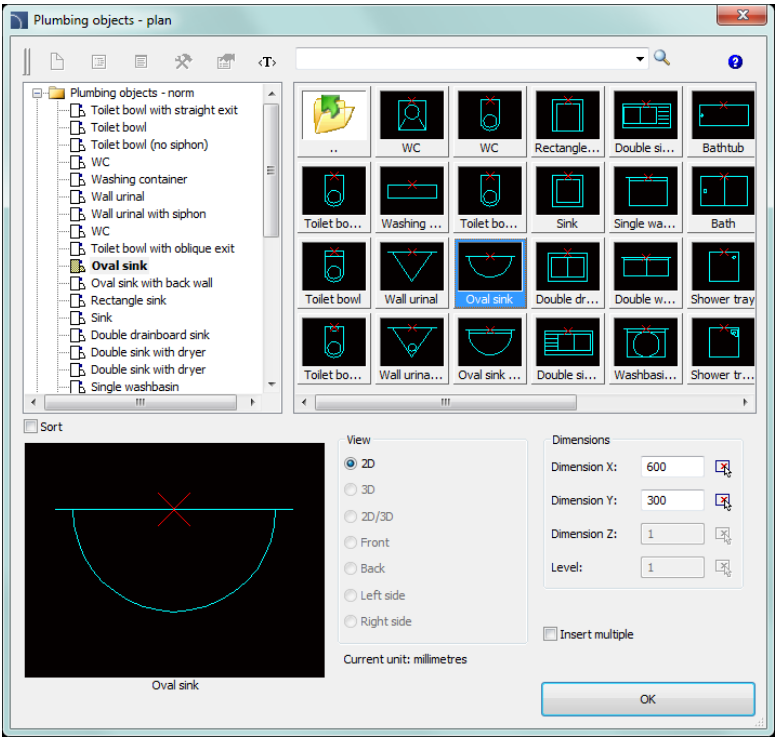


It is also possible to save the user dimensions by Creating and extending the user type of series Creating and extending user type of series (check page 88).

Plumbing objects – plan

The **Plumbing objects - plan** command allows to insert plumbing objects and devices used in designing sanitary installations. Plumbing objects views are prepared in accordance with descriptions used in the respective standards.

Objects inserted into the drawing must have dimensions that correspond with the actual dimensions. Because of that user has the possibility to directly specify objects dimensions before inserting it into the drawing.



Plumbing objects – plan dialog window

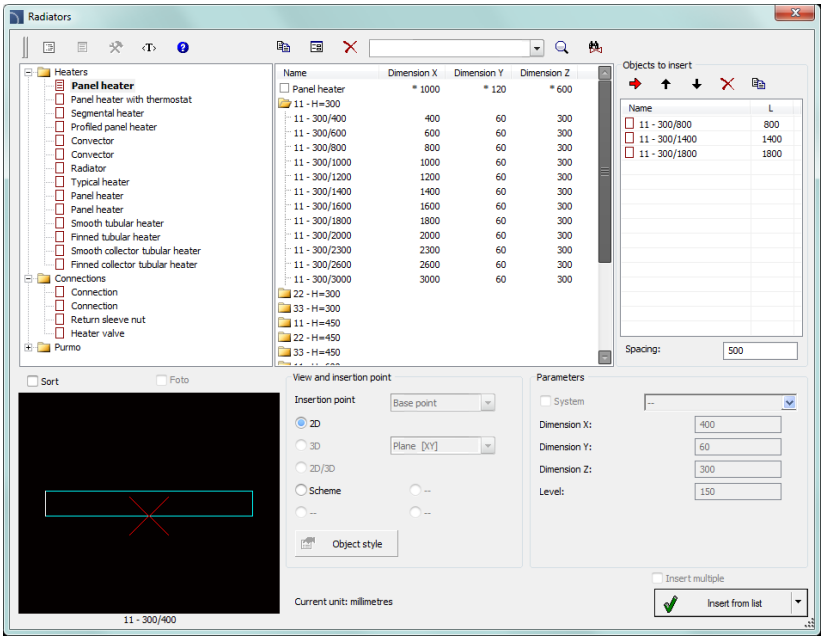
Plumbing objects - plan dialog window contains many options that are described in the Parametrical armature (check page 202) chapter. Additional options that allow to specify objects dimensions:

Types, data, descriptions – possibility to choose the products from the database (this option is available only for some elements).

Insert multiple – enables/disables the possibility of inserting multiple symbols to the drawing.

Radiators

The **Radiators** command is used to insert different types of radiators to the drawing. Views for schemes and 2D plans drawings are available.
Radiators dimensions can be determined by the user (but only for the first element from the list). Other radiators have a specified type, that's why their dimensions shouldn't be changed.
This command gives user the ability to create a list of **Objects to insert**, thanks to which it is possible to add several different radiators simultaneously to the drawing.



Radiators dialog window

Radiators dialog window contains many options that are described in the **Parametrical armature** (check page 202) chapter. Additional options allow to specify objects dimensions and to create a list of inserted radiators:

Objects to insert – list of objects that will be inserted into the drawing. The objects order can be changed with the help of move selected rows up, move selected rows down buttons.
The **Add element to a list** button adds object to the list of objects to insert.

Spacing – specifies the distance between objects that will be inserted.

Procedures

Inserting multiple objects (list of objects to insert)

1. Run the **Radiators** command.
2. In the dialog window select the appropriate category. After that the element preview will be displayed and on the detailed list you will be able to see all available element types and sizes.
3. (Optional) If the list of available types possesses subcategories double click on a category will expand it and you will be able to see its content.
4. (Optional) Right click on any item and from the menu choose the **Expand all/Collapse all** option.
5. Choose the needed element form the detailed list.
6. (Optional) Change the **View** for the selected element.
7. In order to add an element to the list of objects to insert, double click on the chosen element or click the **Add element to a list** button.
8. Continue with filling the list by repeating the steps **2 - 7**.
9. (Optional) Select the element from the list of **Objects to insert** and with the help of the **Copy**, **Delete** buttons, copy or delete a selected element from the list.
10. (Optional) Select the element from the list of **Objects to insert** and with the help of the move selected rows up, move selected rows down buttons change the order of element in the list.
11. Specify the spacing of inserted objects.
12. Click the **OK** button to insert objects into the drawing.
13. Specify the insertion point and the objects rotation angle.

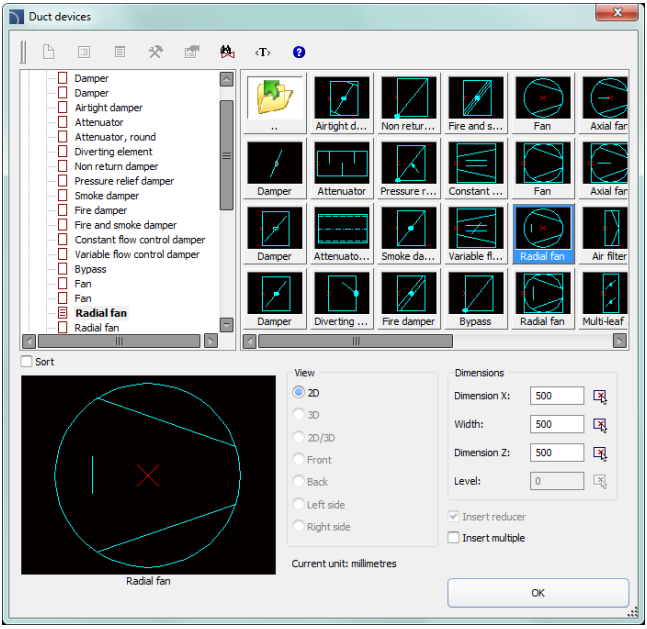
Specifying the dimensions

1. Run the **Radiators** command.
2. In the dialog window select the appropriate category. After that the element preview will be displayed and on the detailed list you will be able to see all available element types and sizes.
3. Select an object from the detailed list. Field that allows you to specify dimensions will be activated.
4. Specify dimensions.
5. Click the **OK** button to insert object into the drawing.
6. (Optional)Click the **Add element to a list** button to add an element with specified dimensions to the list of **Objects to insert**.

It is also possible to save the user dimensions by Creating and extending the user type of series **Creating and extending user type of series** (check page 88).

Duct devices

The **Duct devices** command allows the insertion of duct devices. It is possible to insert element in previously drawn ducts or at any point. When inserting elements in previously created ducts, it is possible to automatically insert necessary reducers.



Duct devices dialog window

Dialog window contains many elements from the **Parametrical armature** (check page 202) chapter. Additional options allow to specify dimensions or insertion settings:

Dimensions – possibility to specify object dimensions.

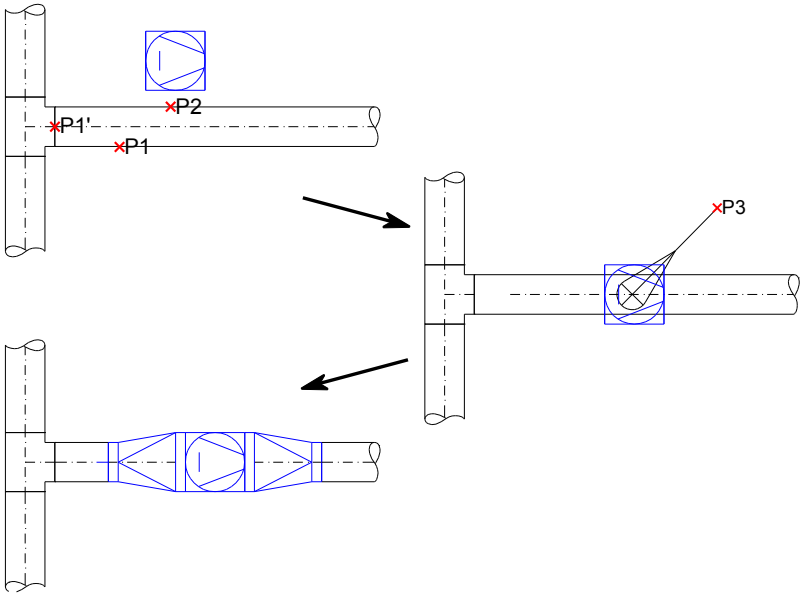
Insert reducer – automatic insertion of reducers when inserting objects into ducts. Reducers are being inserted when the duct width is different than the width of inserted object.

Procedures

Inserting a duct device

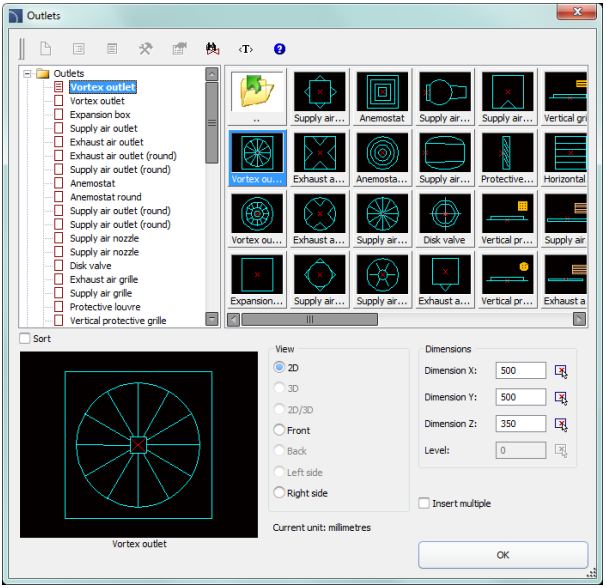
1. Run the **Duct devices** command.
2. Specify the dimensions of object that is going to be inserted.
3. (Optional) Enable/disable the **Insert reducer** option.
4. (Optional) Enable/disable the **Insert multiple** option.
5. Click the **OK** button to insert object into the drawing.
6. Indicate a duct in which you would like to insert an element (**P1**). Application will automatically search and choose the closest duct end as the reference point (**P1'**) and specify the actual insertion point. In order to indicate an insertion point (**P2**) enter a distance measured from the end of chosen line or indicate a point on it. Indicating the point (**P3**) allows to specify the object insertion direction.
7. (Optional) Click at any point in the drawing to add an element without inserting it into the duct.

Duct devices



Outlets

The **Outlets** command allows insertion of outlets, ventilators, exhaust units, fans, air intakes and other ventilation elements.



Outlets dialog window

Dialog window contains many elements from the **Parametrical armature** (check page 202) chapter.

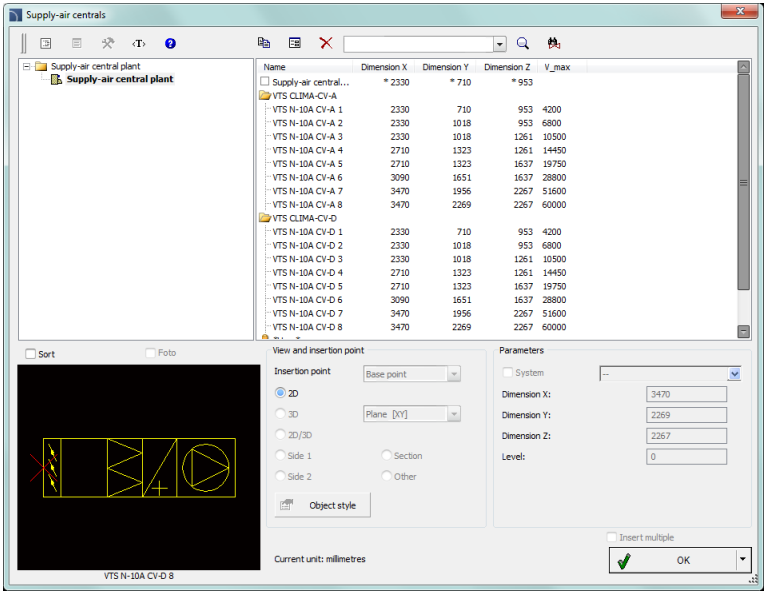
Procedures

Inserting outlets

1. Run the **Outlets** command.
2. Specify the dimensions of object that is going to be inserted.
3. (Optional) Enable/disable the **Insert multiple** option.
4. Click the **OK** button to insert object into the drawing.
5. Specify the object insertion point.
6. Specify the object rotation angle.

Supply – air centrals

The **Supply – air centrals** command allows insertion of air centrals in 2D view.



Supply – air centrals dialog window

Dialog window contains many elements from the **Parametrical armature** (check page 202) chapter.

Procedures

Specifying dimensions for object that is going to be inserted

1. Run the **Supply – air centrals** command.
2. Select the needed element in the dialog window. For the first element from the list the fields that allow to specify dimensions are activated.
3. Specify the dimensions of air central that is going to be inserted.
4. Click the **OK** button to insert object into the drawing.
5. Specify the object insertion point.
6. Specify the object rotation angle.

CADprofi® Electrical

CADprofi Electrical

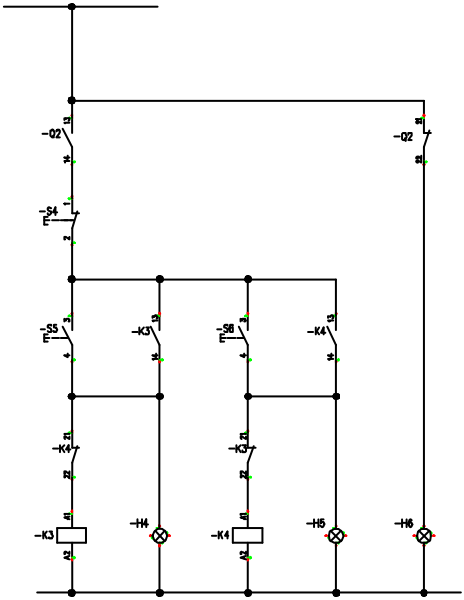
CADprofi Electrical helps with designing any electrical installations, including supply, lighting, low voltage, telecommunications, security, aerial and more. Several thousands of symbols from the latest electrical standards, lighting fixture, switchgear, line drawing system as well as ducts and cable trays are available. Program includes several particularly useful features including the possibility of automatic numbering (addressing) of electrical circuits and a convenient schematic editor.

CADprofi Electrical in 2 minutes

CADprofi Electrical – introduction

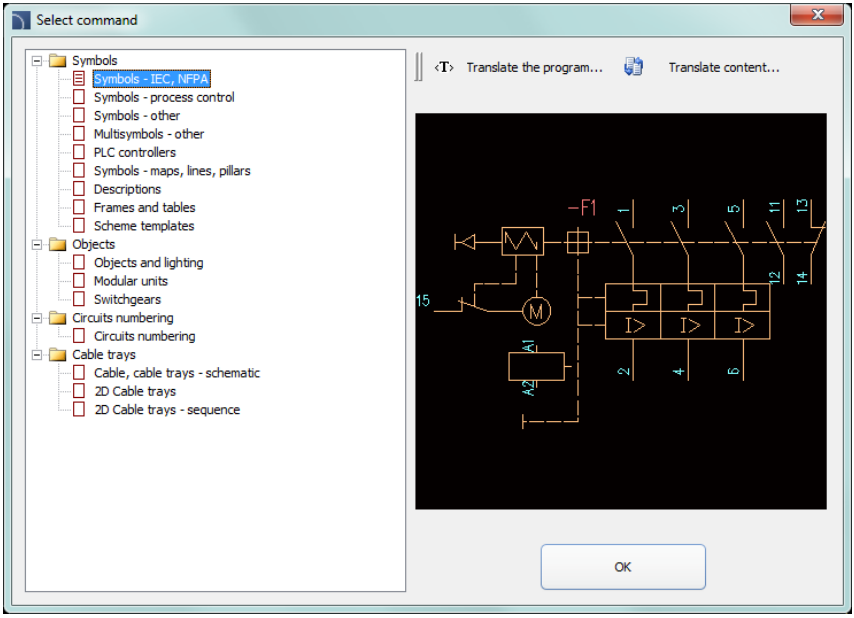
Creating installations is connected with drawing lines, inserting symbols and describing drawings. **CADprofi** program offers the designer appropriate commands that make work easier and faster.

Many options automate frequently performed actions, so it's very important to know how to work in the program. An important stage when creating projects is to describe the project elements, making it possible to create complete BOMs.



Electrical – general commands

The **Electrical** command opens the dialog window from which we can choose the appropriate commands from the **Electrical** module. This command has been prepared for beginners. In the dialog window apart from the list of all commands we can also find a large preview window that shows the outline of the selected function.



Electrical command dialog window

Procedures

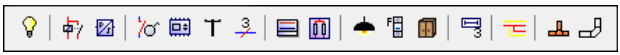
Selecting the command

1. Run the **Electrical** command.
2. In the dialog window, expand the chosen category and select the needed command.
3. Selected command will start after clicking the **OK** button.

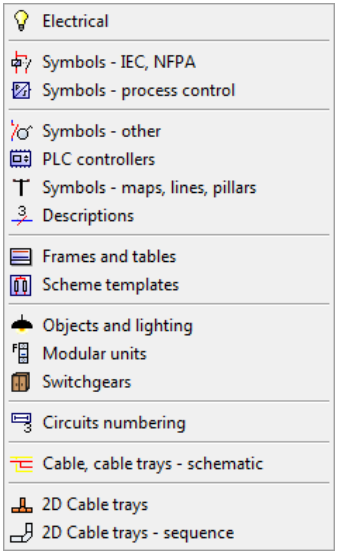
CADprofi Electrical: main program commands

CADprofi Electrical menu and toolbar

CADprofi Electrical commands



CADprofi Electrical toolbar



Menu of CADprofi Electrical module

Editing commands

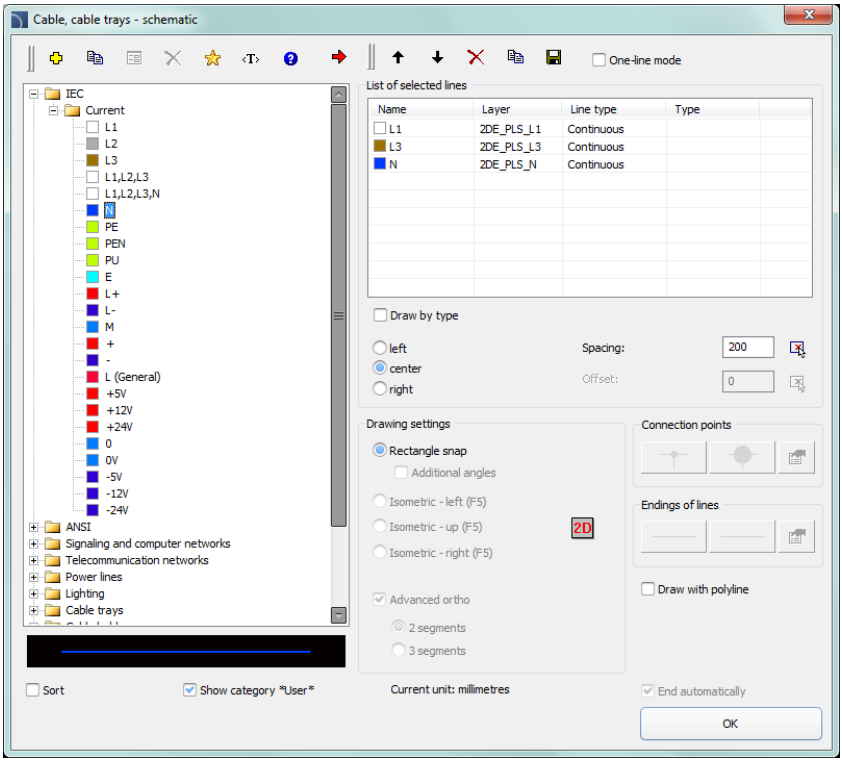
Important editing commands used in designing Electrical installations can be found in the **CADprofi** menu and **CADprofi – Tools** toolbar.

CADprofi – Tools toolbar

Lines, cable trays - scheme

In **CADprofi** program schematic lines are drawn with lines or polylines that lie on the corresponding layers. The term "**line kind**" means the intended use of the selected line or a specific system (for example the **L1,L2,L3** line). It is also possible to add "**type**" to the line, which defines its physical properties (e.g. YDY 3x2,5 for installation lines). Line type can be specified during drawing or later after the lines are already drawn with the **Schematic lines - edit** command.

Standard line kinds from **IEC** and **ANSI** standards are grouped in categories. The **Cable, cable trays - schematic** command gives the possibility to draw one or many electrical installation lines. The process of drawing single lines is possible after selecting in the dialog window the necessary line kind and clicking the **OK** button. This procedure however doesn't allow to change the drawing settings. These options are available only after user has added a line to the list of list of selected lines. User can add many lines to this list, thanks to which it is possible to simultaneously draw several lines.



Cable, cable trays – schematic dialog window

The dialog window contains the following elements:

Lines menu – tools menu that allows user to manage lines' kinds:

- **Define new** – allows to add a new position (line) to the list as a user line. Afterwards it opens a new **edit** window and allows user to edit the line data
- **Copy** – copies the selected line with all its parameters, therefore creating an element made by user. Copied lines may be edited.
- **Edit** – opens the edit properties window of user created lines. Line name can be edited directly by pressing the **F2** key.
- **Delete** – deletes the selected line from the list. Only the lines created by user may be deleted.
- **Add to favourites** – adds the selected line to the ***Favourites*** list.
- **Add element to a list** – adds the selected line to the **List of selected lines**.

List of selected lines – contains lines, which will be drawn after clicking the **OK** button.

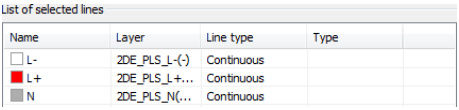
Menu of selected lines list – allows to manage the content of the selected lines list.

- **Move selected rows down** – moves the selected line one position down.
  Move selected rows up – moves the selected line one position up.
- **Remove selected rows** – deletes the selected line from the list.
- **Copy selected rows** – copies the selected line and adds it to the list.
- **Save as set of lines** – allows to save the current list of selected lines as a new set.

> ***Notice***
>
> ***It's not possible to create (save) a set of lines if the list contains at least one line from the *Installations from drawing*.***

- **One-line mode –** this option blocks the possibility of adding more than one line to the **List of selected lines.** When enabled, program will convert the list of lines to a one line, chosen by user.

Type of line – it is possible to define **Type** for each line.



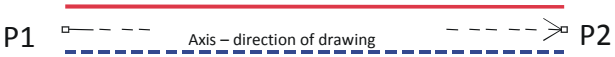
In order to do this, user has to enter an appropriate value in the **Type** field that is located next to the line name on the **List of selected lines**.

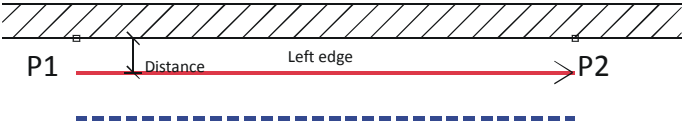
Draw by type – enables the **line type** view for the newly created lines.

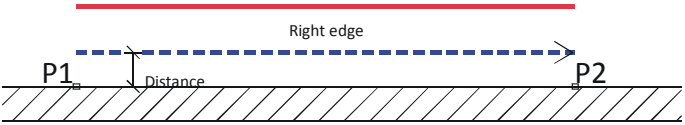
Lines conducting options – options that allow user to specify the way of drawing lines.

- **Left** – draws lines on the left side of the indicated points.
- **Center** – draws lines symmetrically regarding the indicated points.

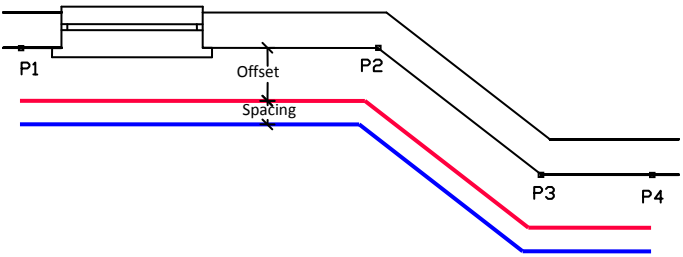
- **Right –** draws lines on the right side of the indicated points.
- **Spacing –** the distance between the lines.
- **Offset –** an offset of the drawn lines from the points that are actually indicated in the drawing. This option allows to easily draw lines at a specified distance along the walls or other elements that exist in the drawing.







> ***Hint***
>
> ***Thanks to combining both the drawing edge and the offset user can lead the lines in accordance with any existing objects that are already inserted on the drawing e.g. walls.***



Show category *User* – enables/disables the visibility of the folder that contains all categories and lines created by user. If you turn off this feature then lines defined by user will be "mixed" with other standard **CADprofi** lines.

Drawing settings – line drawing options.

- **Rectangle snap** – 2D drawing default settings.
- **Additional angles** – enables the diagonal dimetric mode, through the use of polar tracking options (POLAR) according to the angle value that was defined in the **CADprofi – options** window.
- **Isometric** – allows to turn on the mode that allows to create isometric schemes (SNAPSTYLE variable). Three options are available (left, up, right) which can be easily switched during drawing isometry by pressing the **F5** key.

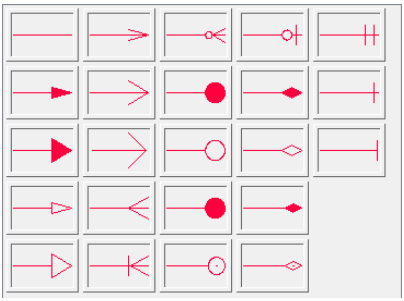
Connection points – when drawing a single line it is possible to automatically insert connection points. There are several options available:

- **Lack** – no symbol on the line joint.
- **Always** – adding a symbol regardless of the kind of connected lines.
- **On the same layer** – adding a connection symbol only when both lines are on the same layer.

> ***Hint***
>
> ***If user uses an extended layer structure this option will add a connection symbol only on lines with the same type.***

- **Connection symbol** – allows to select the graphical symbol that will be inserted in the place where lines connect. User can select one of the following symbols:

- **Settings** – setting up the **scale** (symbols' size) and the **Remove unnecessary connection points** option, which automatically searches for unnecessary connection points (e.g. remnant after line removal) and removes them from the drawing.
- **Ending of lines** – allows user to select symbols that will be inserted in the beginning and ending of lines.

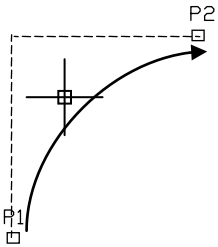


- **Settings** – setting up the **scale** (symbols' size) and the **Remove unnecessary connection points** option, which automatically searches for unnecessary symbols of line endings (e.g. remnant after line removal) and removes them from the drawing.

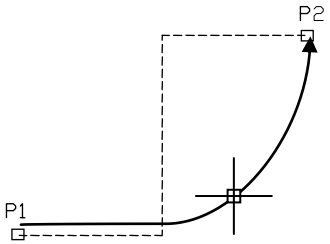
Advanced Ortho – in this mode between the indicated points, two or three orthogonal segments are being drawn. This mode automates the process of connecting objects or lines while maintaining a right angle between the segments.

- **2 segments** – creates a line consisting of two segments.
- **3 segments** – creates a line consisting of three segments.

> ***Notice***
>
> ***Advanced ortho mode is available only in certain CAD programs.***
>
> ***The first drawn segment can be vertical or horizontal depending on which way of the mouse movement is performed after selecting the first point (P1) of the drawn line.***



Advanced ortho 2 segments

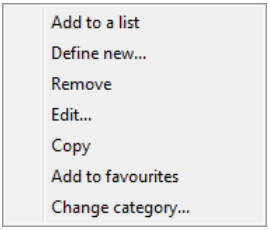


Advanced ortho 3 segments

Draw with polyline – enables/disables drawing line segments as polyline.

End automatically – enables/disables automatic ending of lines when the drawn line is being connected to a different object in the drawing.

Pop-up menu – options menu for selected element/category which is available by clicking the right mouse button. Commands included in the menu are similar to the commands available on the dialog window toolbars.



Procedures

Drawing schematic lines

1. Run the **Cable, cable trays - schematic** command.
2. (Optional) Select the line and click the **OK** button in order to allow the drawing of a single line without making changes in the settings.
3. In the newly opened dialog window pull-down the category that contains the desired line kind.
4. Select the desired line and click the **Add element to a list** button, or double click on it in order to put it on the **List of selected lines**.
5. (Optional) Repeat steps 2 and 3 in order to add more lines to the list.
6. (Optional) Using the **Copy selected rows**, **Remove selected rows** buttons add or remove the selected line from the list.
7. (Optional) Using the **Move selected rows down**/**Move selected rows up** buttons change the order of the lines on the list.
8. (Optional) Fill the information about type of line, enable/disable the **Draw by type option**.
9. (Optional) If there is more than one line on the **List of selected lines** specify the **Spacing** value.
10. Specify the way how you will draw the line and specify the **Offset** value if needed.
11. (Optional) If on the **List of selected lines** there is only one object then change the **Drawing settings**.
12. (Optional) If on the **List of selected lines** there is only one object you can set the **connection** and **line endings** options.
13. Enable/disable the **End automatically** option.
14. Click the **OK** button in order to start drawing a line (or several lines).
15. Specify the starting point and next segment points that you want to draw. In order to finish click the **Enter** key.

> ***Notice***
>
> ***Offset, additional angles, adding connection points, endings of lines, advanced ortho and end automatically options are available only when a single line is selected.***

Schematic lines are simple lines or polylines that's why it is also possible to draw lines with the CAD program standard commands such as **Line** or **Polyline**. However, it should be noted, that lines drawn in this way should be placed on appropriate layers. In order to match

properties (e.g. layers), between the drawing objects, the **Match properties** (_MATCHPROP) command located in the CAD program can be used.

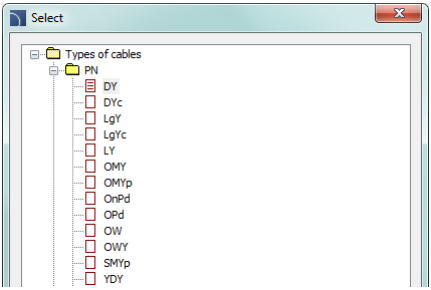
> ***Hint***
>
> ***The „Schematic lines – edit" command is used to edit type of lines (check page 67).***

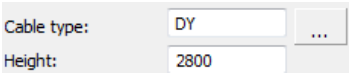
Drawing risers (vertical lines)

In schematic lines it is possible to present vertical lines segments using special symbols, such as **risers**.

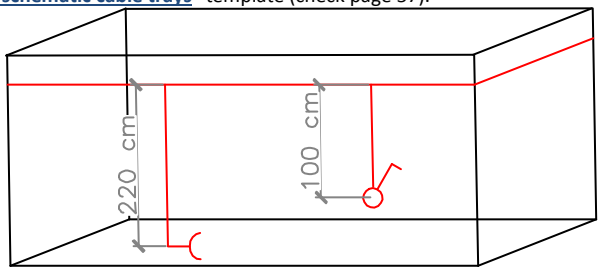
Despite the fact that **risers** are being inserted using the **Symbols - other** (check page 251) command, program treats them as **schematic lines**.

For risers user should specify the type of line and height (length of vertical line segment). Line type can be selected from the list that will be available after pressing the **Browse** ... button. Thanks to this, their diameter data and length are being included in BOMs.

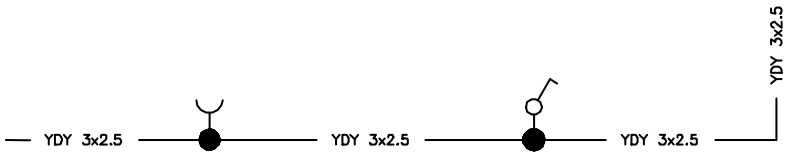




Data that is saved in the riser is included during the BOM creation when user uses the „**Electrical – schematic cable trays**" template (check page 57).



Installation view



Scheme with risers view

Name	Line type	Duct pcs	Sum	--
Line	YDY 3x2.5	1	2.30	m
Line	YDY 3x2.5	1	1.76	m
Line	YDY 3x2.5	1	1.74	m
Line	YDY 3x2.5	1	2.20	m
Line	YDY 3x2.5	1	1.00	m

Lines BOM view

Linetype scale for lines

Each line is being drawn with different line patterns, such as a dashed line (DASHED) used for the "Return" line. The line pattern density is set using the CAD command: LTSCALE (_LTSCALE). By default new drawings have got a linetype scale factor set as 1.00. If this factor number is too small or too big then line pattern may not be visible and all lines will be seen as drawn with a continuous line.

LTSCALE = 200

LTSCALE = 50

In **CADprofi** program it's possible to design in units set as millimeters, centimeters or meters. For this reason, when drawing schemes the linetype scale should be always adapted to the edited scheme. For example, when drawing a scheme with the unit set as "millimeters" with symbol's size set as 200 then the linetype scale factor should be between the 50 and 200 range.

Defining new user lines

User can add his own type of lines to the program. In this way, program can be adapted to user's needs and can be used in a variety of projects in which it is necessary to draw schemes.

Procedures

Adding new lines in the database

1. Run the **Cable, cable trays - schematic** command.
2. Click the **Define new** button in order to add a new line to the existing database. New element will be added to the ***User*** category. A new **Edit** window will open, in which user will be able to specify all line parameters.
3. (Optional) Select an existing line and click the **Copy** button to create a new item. Selected line will be used as a template.
4. Change the new line settings in accordance with the **Edit line data** procedure.

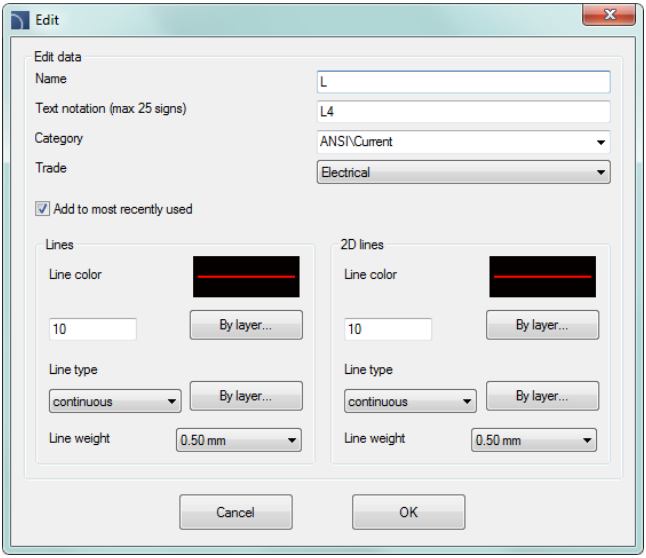
***Favourites* category**

It is also possible to create a folder that contains a list of most frequently used installations by the user. This folder is created automatically when a first element marked as **favorite** has been added to the program. In the ***favorite***category user can define any branch structure, in which the individual elements will be located.

Edit line data

CADprofi allows to edit parameters (drawing features) of lines created by user. To do this, use the **Edit** option.

In **CADprofi** program pipe and ventilation lines can be drawn as schematic (with use of lines or polylines) or in 2D view. Therefore, when editing properties it is possible to specify different parameters for schematic lines as well as for lines in 2D view.



Edit line data dialog window

The **Edit** dialog window contains the following items:

Name – name of edited line.

Text notation – text marking for lines used as an ending of layer's name in the extended layer structure.

> ***Notice***
>
> ***It is recommended to use „unique values" of text notations in order to ensure proper recognition of lines by CADprofi.***

Category – currently assigned category in **CADprofi** database.

Branch – currently assigned industry branch in **CADprofi** database.

Add to most recently used – this option adds the edited line into the default list of line types in the in the **2D Pipes – plan** dialog window.

Lines or **2D lines** – settings used for schematic lines or lines drawn in 2D:

- **Line color** – determines the line color. Color can be selected in several ways: by clicking on the button that contains the line preview, with the help of the **by layer** option or by entering CAD program color number.
- **Line type** – determines the line type (kind). Line type can be selected from the list of available line types or from the **by layer** option.

- **Line weight** – determines the line width. Line width can be chosen from the list available values in the program.

> ***Notice***
>
> ***Only lines defined by user may be edited (it is not possible to change line parameters from the primary CADprofi database).***

Procedures

Edit user lines

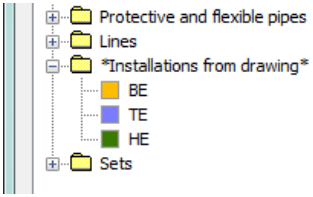
1. Run the **Cable, cable trays - schematic** command.
2. (Optional) If the line that you would like to edit is located on the list of selected lines, then select it and click the **Remove selected rows** button in order to remove it from the list.
3. On the **List of available lines** select the line that you would like to edit.
4. Click the **Edit** button A new dialog window will open, allowing you to edit the line.
5. (Optional) Change the **Name** of edited line.
6. (Optional) Change the **Text notation** of edited line (not recommended if the edited line is already inserted in the drawing).
7. (Optional) Change **Category** of edited line.
8. (Optional) Change the **Trade** for selected line.
9. (Optional) Enable/disable the **Add to most recently used** option.
10. (Optional) Change parameters of schematical **Lines**.
11. (Optional) Change parameters of schematical **2D Lines**.
12. Click the **OK** button to accept the changes.

> ***Notice***
>
> ***If you have changed the parameters for lines that have already been added to the drawing then in order to update their properties you will need to insert a new line (that has the new parameters) into the drawing.***

Installations from drawing

All lines and polylines that lie on appropriate layers are treated by **CADprofi** program as schematic lines. This includes layers with names similar to **2DX_PLS_xxx**.

When several designers are working on the same project, then it is possible that one of these designers has used his own line kinds and then has forwarded the drawing to further edition. In this case, **CADprofi** application will automatically detect the presence of additional lines (installation kinds) and will display them in a new category called **Installations from drawing**. Name of the new lines that were identified from the drawing are being taken from their label (ending of the layer's name). If these lines' parameters will be edited in the dialog window, then they will be permanently saved in the program's database as user's lines.



It is possible to use **Installations from drawing** without the need of editing them, but in this case, on all BOMs the **Installations from drawing** lines will have their names listed as their layers' label.

Lines' layers

Each line kind is being drawn on separate layers. In order to easily manage layers in the top menu of **CADprofi** programs are located additional commands that allow to quickly hide or show all layers that belongs to central heating, sewage, water and other kinds of installations. In the *C:\Cadprofi\X.x\PlotStyle* printing styles are located that suit the program's layers structure. Out of these files user can select monochrome printing (*CADprofi Mono.ctb*) and color printing (*CADprofi Color.ctb*).

*.ctb files should be copied to the appropriate folder of used CAD program.

For more information, please refer to the **layers section** (check page 32).

Working with symbols – general rules

CADprofi program possesses several commands used for inserting symbols into the drawing. It is possible to insert symbols in any point, into lines, as well as the simultaneous insertion of multiple symbols in the drawing.

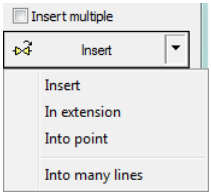
CADprofi symbols have certain properties, thank to which they behave in a "smart" way, for example automatically adjust the insertion angle to the line, automatically break the line at insertion point, adjust the layer depending on the kind of installation etc.

All symbols available in **CADprofi** program are being inserted as blocks, which can be copied, deleted or edited with the use of standard CAD program commands. Most symbols can be also edited with the use of **CADprofi** editing commands such as: **Quick edit**, **Edit symbols** and **Delete symbols**.

Symbols are often used to graphically represent real devices, armature and other installations elements, therefore it is possible to add technical parameters, about the products used in the project, to the symbols (**Attributes and descriptions** command), as well as numbering and symbols' marks. This information can be used when describing drawings when creating graphical **legends or specifications** (check page 46).

Symbols and **Marks** commands are used to insert symbols into the drawing.

Each command that is used for working with symbols has got a variety of options used insertion of single or multiple symbols into the drawing.

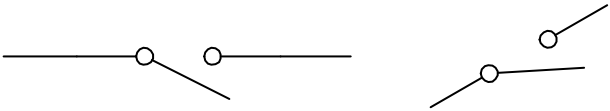


Main options (available after pressing the button):

- Insert symbol or symbols.
- Into many lines.
- In extension.
- Into point.

'Insert symbol' option

This is the default option that allows to insert symbols in any point or into a line indicated by user.



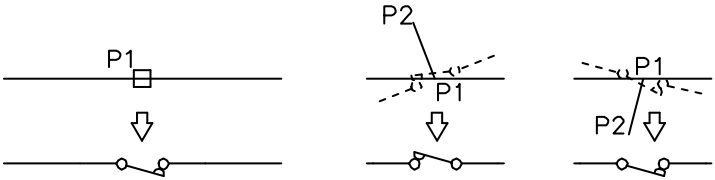
Insert into a line *Insert in any point*

Procedures

Insert symbol

1. Select the symbols or symbols in any library.
2. Click the **Insert symbol** button. The library dialog window will close allowing insertion of symbols into the drawing.
3. Point into a line, in which you would like to insert the symbol.
4. (Optional) Click anywhere in the drawing to insert a symbol.
5. (Optional) Specify the rotation angle of inserted symbol - this option is available only if user has selected the **Rotate** option in the library dialog window.

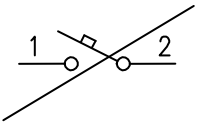
When inserting symbols into lines, the symbol insertion angle depends on the line angle (the angle specified in the dialog box is being ignored).
In this case, if user selects the **Rotate** option then he will be able to specify symbol's rotation relatively to the line. This option is very convenient when inserting symbols which have a possibility to determine their tap direction, handle position, flow direction etc.
The angle of such symbol will be dependant on the position of **P2** point.



Without rotation *With rotation*

'Into point' option

The **Into point option** may be used when it is necessary to insert the symbol in a point that lies on a line, but in such a way that the symbol won't "adjust itself to the line". In this option, the inserted symbol "ignores" the line, so it does not break it and it does not take angle from it. This option is used in many situations for example when inserting sensors, which usually need to touch lines.



Inserting symbol in a point (line was ignored)

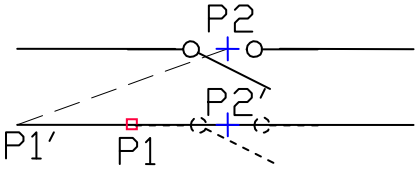
Procedures

Insert into indicated point

1. Select the symbols or symbols in any library.
2. With the use of button, pull-down the insertion list.
3. Click the **Into point** button. The library dialog window will close allowing insertion of symbols into the drawing.
4. Point into a point in a line, in which you would like to insert the symbol.
5. (Optional) Click anywhere in the drawing to insert a symbol.
6. (Optional) Specify the rotation angle of inserted symbol - this option is available only if user has selected the **Rotate** option in the library dialog window.

'In extension' option

The **In extension** option allows to precisely specify the symbol insertion point in relation to other objects located in the drawing. This allows to insert a symbol in a point in accordance with for example another symbol.



Inserting in extension

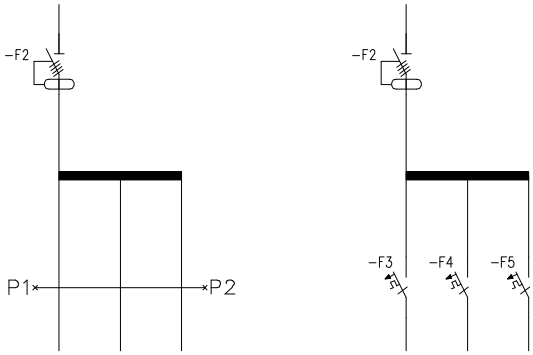
Procedures

Insert in extension

1. Select the symbols or symbols in any library.
2. With the use of button, pull-down the insertion list.
3. Click the **In extension** button. The library dialog window will close allowing insertion of symbols into the drawing.
4. Click a line, in which you would like to insert the symbol (**P1**).
5. Program will determine a point at the line beginning (**P1'**). Specify the offset against this point or specify the symbol insertion point against you would like to specify the location of the inserted element (**P2**).
6. Specify an additional offset or click **Enter** to confirm the insertion point.
7. (Optional) Specify the rotation angle of inserted symbol - this option is available only if user has selected the **Rotate** option in the library dialog window.
8. Symbol will be inserted in the drawing and the command will end (if user has not selected the **Insert multiple** option).

'Into many lines' option

Into many lines option gives the possibility to quickly insert selected symbol into many lines. It allows to insert selected symbol into all lines that cross with the „indication line", that was specified during insertion (**P1–P2**).



Inserting symbols into many lines

Procedures

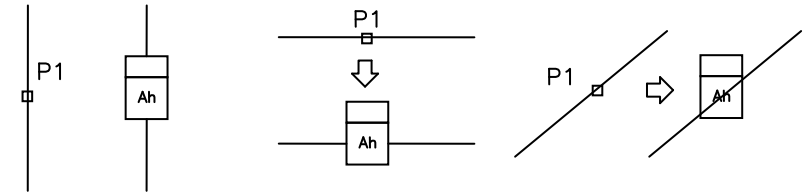
Insert into many lines

1. Select the symbols or symbols in any library.
2. With the use of button, pull-down the insertion list.
3. Click the **Into many lines** button. The library dialog window will close allowing insertion of symbols into the drawing.
4. Specify the first point of the "crossing line" that will cross with lines (**P1**).
5. Specify the second point of the "crossing line" that will cross with lines (**P2**).
6. The selected symbol will be inserted in all lines that crossed with the "crossing line".

Schemes: working with symbols

Usage of symbols inserted with a fixed angle

In order to maintain the project correctness, some symbols are being inserted with a fixed angle. Examples include gravels and separators. These symbols should be inserted with a 0° angle. In **CADprofi** many symbols have got an enforced insertion angle. When user will insert such symbols in horizontal or vertical lines, these lines will automatically break. In other cases, lines are not being broken.



Vertical line *Horizontal line* *Incline line*

> ***Notice***
>
> ***For symbols with a fixed insertion angle, "Angle" and "Rotation" options located in the dialog box are being ignored.***

Symbols that contain text (visible attributes)

For symbols that contain text (visible attributes) **CADprofi** program will automatically adjust angle of the text. Thanks to this the situation in which text would be displayed "upside down" or "backwards" is being avoided. User has the following possibilities to adjust texts:

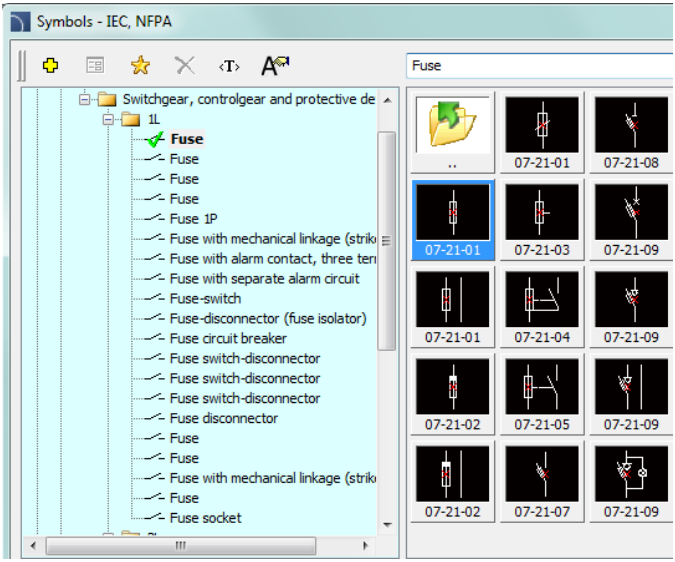
- **0° fixed angle**.
- **0° or 90° angle**. If symbol is inserted with a 90° angle, then the text is also rotated by 90°. In other cases the text angle is 0°.
- **Automatic angle** from -30° to +120°. In these ranges the text rotation angle is sufficiently readable.

Adjustment of attributes angle

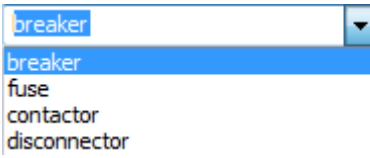
> ***Notice***
>
> ***Options for text adjusting are permanently set for each symbol. User can change these options only for symbols that were added by the user.***

Find symbols

In each dialog window that is used for symbols insertion there is a search option, thanks to which it is possible to quickly find desired symbols.



The **Find** field is used to enter a phrase (full or just a part name of searched symbol). This field remembers typed words, which allow user to search again from the expanded list by pressing the ▼ button and clicking **Find**..



After filling the search field, application will automatically filter the current library content and display only those categories which contain the objects that match the search results. Clicking the **Restore** button ✖ will display all library objects and end the search.

Multivariant symbols

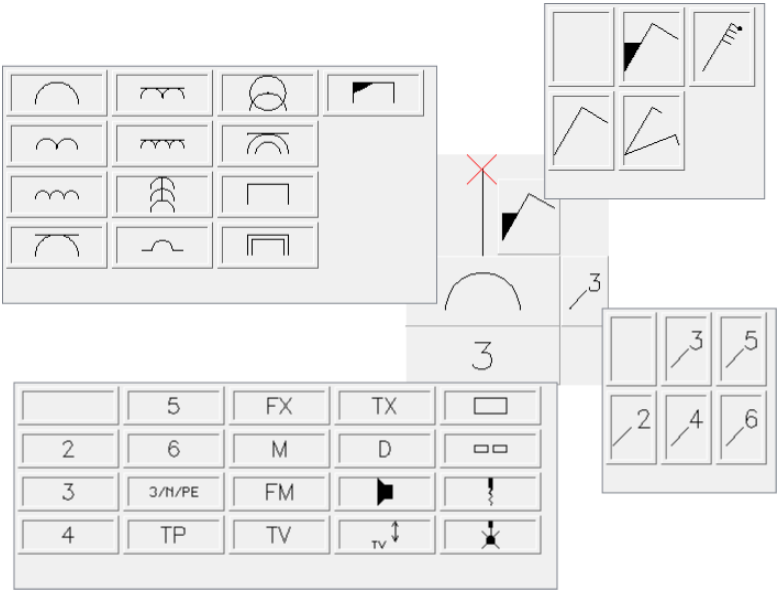
Multivariant symbols are objects, thanks to which in a quick and easy way user can customize their view or information contained in the inserted object. In this way, one multivariant symbol can replace many "traditional" ones, while giving a better possibility to configure their look and type.

Multivariant symbols may be freely modified, using the **Edit symbols** command.

Example of a multivariant symbol

Socket – user can specify the following options:

- socket type,
- connector type,
- socket multiples,
- additional markings (number of fields, type etc.).



Management of symbols content

CADprofi program possesses several thousands of symbols grouped in appropriate categories. Witch such a large number it is sometimes hard to find symbols needed at the moment. On the other hand, when working on a particular project it is usually not needed to work with more than several dozens of symbols. In order to speed the work, user can add selected items to the ***Favourites*** category thanks to which he will be able to get a much faster access to them.

It is possible to create sub-categories related to a specific theme, such as a separate category for symbols used in designing telecommunication, lighting installation and more. Furthermore one element can be added to many sub-categories.

For elements placed in the ***Favourites*** category it is possible to change their name, which will be later used in BOMs, legends and descriptions. Change of name can be also used to user own translations for some symbols from international standards.

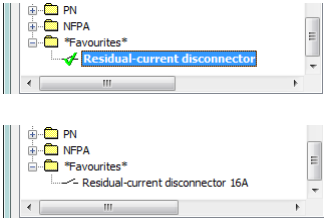
Adding symbols to favourites

Add to favourites button allows to copy any symbol to the ***Favourites*** category. This folder is created automatically after user has copied the first symbol.

Changing the symbol name

Initially, each copied symbol receives the default name from the original item.

In order to change the symbol name that is located in the ***Favourites*** folder, user should first select it and then press the **F2** button to activate the edition mode

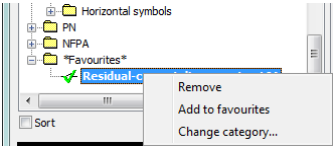


Notice

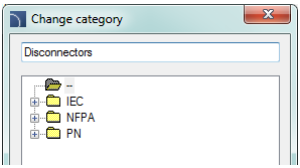
Changed name will be visible in tables and descriptions that are created with the „Bill of materials” (check page 46) command.

Changing category

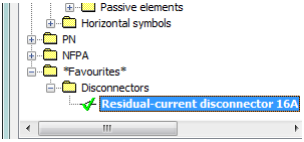
In order to change or define a new category, click the right mouse button on a desired symbol and from the context menu choose the **Change category** option.



In the dialog box, user can select an existing category or enter a new category name. After accepting, selected symbol will be moved to the defined category.

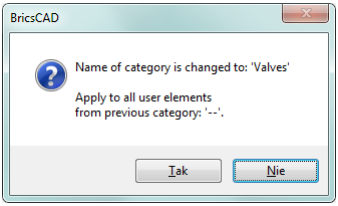


CADprofi application allows to create many copies of each symbol and it is possible to place them in different categories. Thanks to this user can create categories that group symbols of a specified type or the most common used symbols for a given industry type.



Changing the category or many symbols

If user is editing a symbol in his native category in which there are many symbols, then **CADprofi** application will change category for all these symbols.



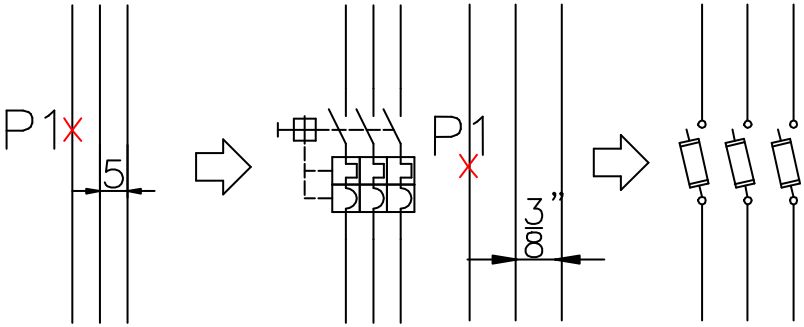
Deleting symbols

It is possible to delete symbols from the ***Favourites*** category. To do so, select the symbol and choose the **Delete** option from the dialog window.

IEC, NFPA symbols

The **Symbols - IEC, NFPA** command allows to insert controlgear, switchgear, complex apparatus, measuring instruments and many additional symbols and elements that are used in electrical installations scheme. In the library there are symbols from both **IEC** and **NFPA** standards available.
During the symbol insertion it is possible to automatically enumerate objects and determinate connections as well as add information about additional technical parameters.

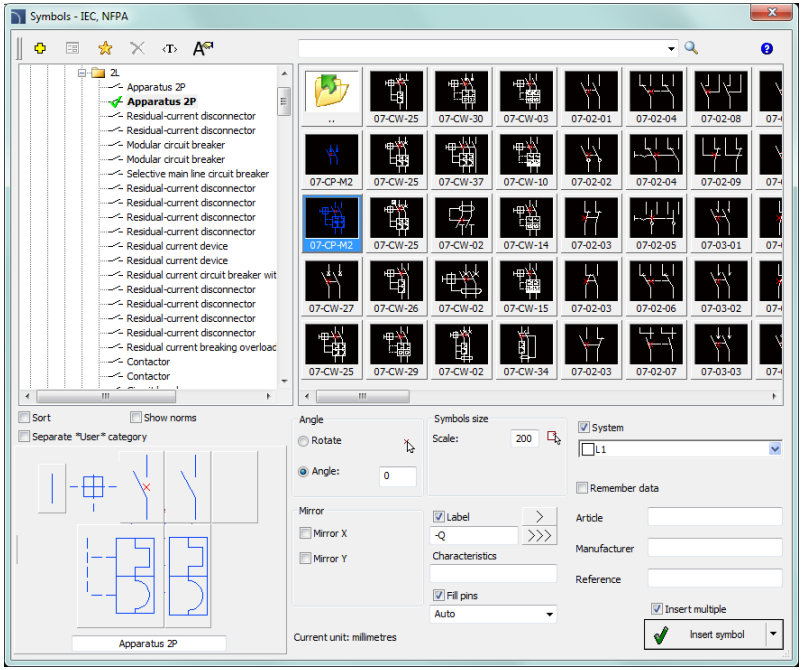
Inserting symbols into lines will automatically interrupt them in the insertion point, while maintaining the principles of creating schematic drawings. This rule also applies to multipole apparatus, but in this case, user has to match the line to the type and scale of multipolar symbols.



Inserting an IEC multipole symbol *Inserting a NFPA multipole symbol*

For **IEC** symbols the offset between each lines is 5 (or a multiple of 5), whereas for **NFPA** symbols it is 3/8" (0.375).

> ***Hint***
>
> ***Templates for potential lines and line circuits that are available in the „Frames and tables" (check page 260) command have been prepared in such a way allow the correct insertion of multipolar symbols, on condition that they have the appropriate scale.***



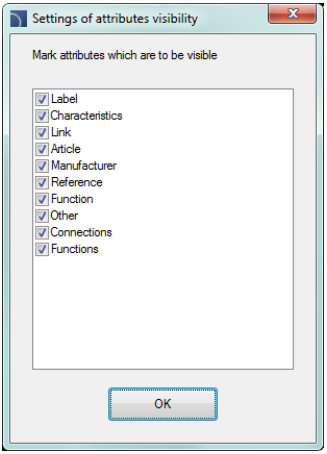
Symbols – IEC, NFPA dialog window

The **Symbols - IEC, NFPA** dialog window contains the following items:

Attributes visibility – this option is used to specify which attributes should be visible when inserting symbols.

After clicking the button, a dialog window will open with a list of available attributes.

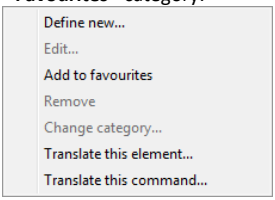
Editing the attributes can be performed after the symbol insertion by using the **Attributes and descriptions** (check page 34) command or by using the DDEDIT (_DDEDIT) command in the CAD program.



Tools menu:

- **Define new** – allows to add a new element to the database. Read more about it in the Defining user blocks (check page 89).
- **Edit** – allows to Editing user block parameters (check page 92).
- **Add to favourites** – copies symbol to the *Favourites* (check page 242).
- **Delete** – deletes the selected symbols from the ***Favourites*** category.

Pop-up menu– options menu for the selected symbol, which is available after right clicking on the mouse button on the selected symbol. Commands in this menu are similar to the tools menu.



Categories tree – a set of all categories and symbols in the library, shown as a pull-down tree. User can customize the display order by using the following options:

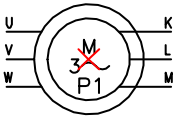
- **Sort** – alphabetical order of categories placement and their content.
- **Show norms**– enables/disables displaying symbols according to norms.

Thumbnail view – displays the content of currently selected category as thumbnails.

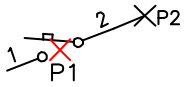
Preview – displays the preview of the currently selected symbol. Right-clicking on the preview window will zoom the view.

Mirror X, Y – enabling this option will create an X or Y mirror image for the specified object.

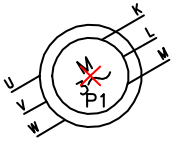
Angle/Rotation – possibility to specify a fixed angle when inserting symbols or enabling the rotation option, in which symbol's angle is defined by rotation during insertion During insertion of symbols into lines, the insertion angle depends on the line angle (the angle specified in the dialog window is being ignored). When inserting symbols into lines it is possible to use the **Rotation** option in order to dynamically specify symbol rotation relatively with the line.



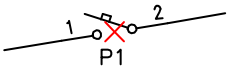
0° angle



Rotation



30° angle



Inserting into a line without rotation

Scale – specifies the size of inserted symbols. It is possible to enter the scale value or to indicate a symbol on the drawing whose scale user would like to use.

System – if in the program options, the extended layer name structure (check page. 29) is enabled then symbols will be inserted on layers depending on the kind of installation (system). In case of inserting symbols into an existing line then the **System** option is being ignored and the symbol layers depends on the line kind.

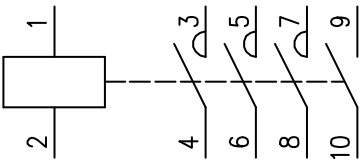
Object data – this option allows to specify symbols technical or identification parameters. For each symbol, different parameters can be given such as: **Article**, **Manufacturer**, and others. Available options and parameters depend on the symbol type.

Remember data– this option allows to save the user entered parameters. This option should be enabled when user inserts symbols with similar characteristics.

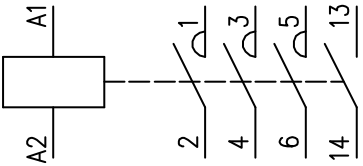
Label – gives the possibility to add subsequent labels to inserted symbols. Label field is automatically filled with default marking and the first free number. Thanks to the available buttons, user can choose between the first free number > and a subsequent number >>>. This field can be also filled with any value.

Fill pins – numbering of connections in the inserted symbol. Numbering type can be chosen from the drop-down list:

- **Auto** – numbering of all connections, starting from number 1.
- **1/2…** or **a/b…** – allows to possibility to manually number connections. This option allows user to enter values himself for subsequent connections, separating them with the"/" sign.



Automatic numbering



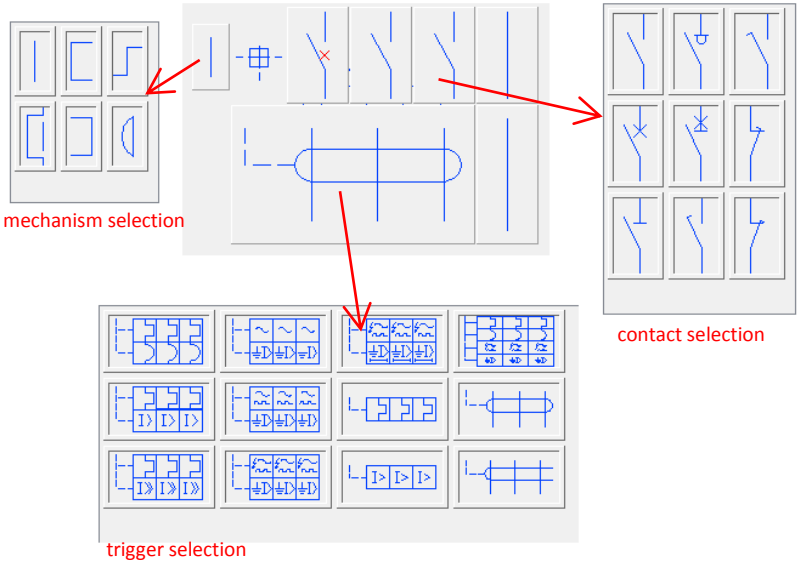
Numbering: A1/A2/1/2/3/4/5/6/13/14.

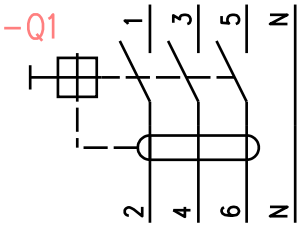
> ***Hint***
>
> ***Entering the "//" characters in the numbering field will allow to bypass the next connection.***

Insert multiple– enables/disables the possibility of inserting multiple symbols to the drawing. The insertion process requires from the user to press the **Enter** or **Esc** key when he finished inserting multiple symbols.

Some apparatus symbols are multivariant symbols, thanks to which user can insert to the project apparatus that contain any configuration of contacts, triggers etc. Such symbols

substitute many "traditional" symbols in the library. In this way it's easier to use the program, and at the same time it gives user more opportunities to define custom elements. Multivariant symbols can be edited with the **Edit symbols** command.



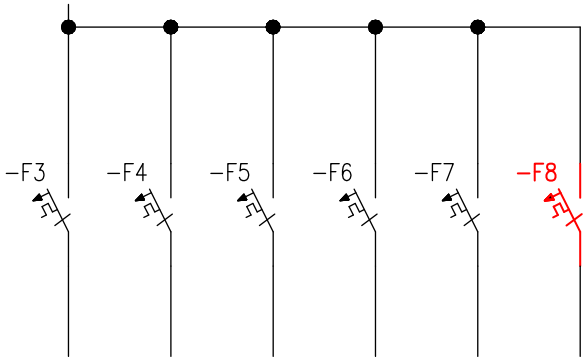


Example of a multivariant symbol

Procedures

Inserting symbol

1. Run the **Symbols - IEC, NFPA** command.
2. In the **Symbols - IEC, NFPA** dialog window choose the appropriate symbol. In the bottom-left corner of the dialog window a preview of the selected symbol will be displayed.
3. Specify the insertion parameters: **System**, **Rotation**, **Scale**.
4. Enable/disable the use of a **Label**, specify the label element, e.g. **–F8**.
5. Enable/disable the **fill pins** option, specify the type.
6. (Optional) Fill out the value of following parameters: **Article**, **Manufacturer**, **Reference**.
7. Click the **Insert symbol** button in order to specify the symbol insertion point.
8. (Optional) With the use of the button pull-down the insertion list and choose **In extension** to specify the insertion point in accordance to other objects.
9. (Optional) With the use of the button pull-down the insertion list and choose **Into point** to insert the symbol in a specific point.
10. (Optional) With the use of the button pull-down the insertion list and choose **Into many lines** to insert symbols simultaneously into many lines.
11. Click on a point in the line to insert the symbol.
12. (Optional) Click on any point in the drawing, to insert the symbol without inserting it on a line.
13. (Optional) Specify the inserted symbol angle - this option is available only if the **Rotation** option in the library dialog window has been enabled.

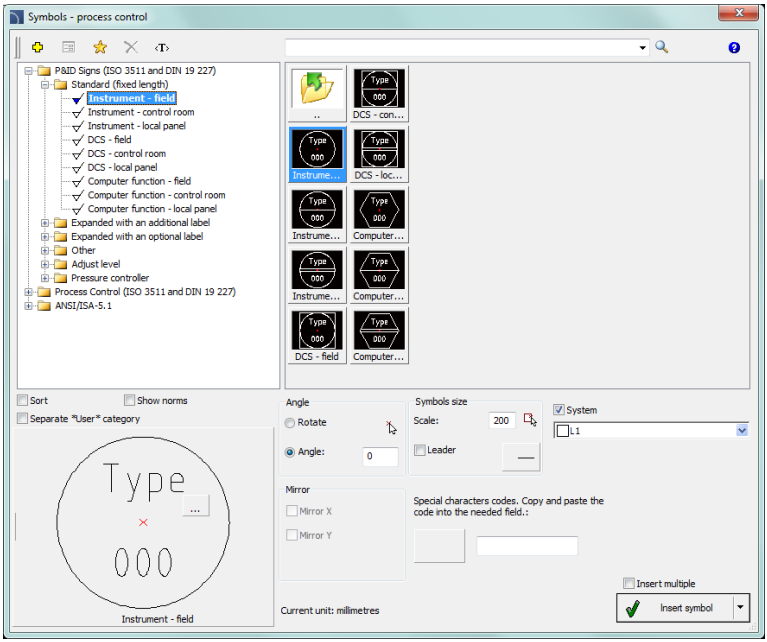


Schemes: working with symbols

Process control symbols

The **Symbols - process control** command allows to design control-measurement systems and industrial automation by using multivariant symbols such as measuring points, sensors, transmitters, controllers etc.

Each, individual points (elements) of the scheme can be automatically numbered. They are also used to determine various markings.



Symbols – process control dialog window

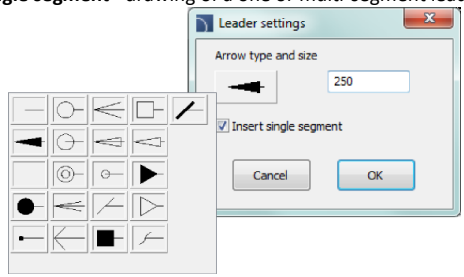
Basic functionality and options regarding symbols are described in the **Symbols - IEC, NFPA** (check page 244) chapter. Additional options for the **Symbols - process control**:

Numbering and marking fields – for each symbols different fields are available, that are used to add additional markings, symbols variants or numbering.

Special symbols codes field – gives the possibility to display the code of the selected symbol so it can be copied and pasted anywhere into a text field.
List of symbols is available after pressing the button.

Leader – enables/disables usage of leader for descriptions. By clicking on the button, that symbolizes the currently set arrow type, user can change the following leader settings:

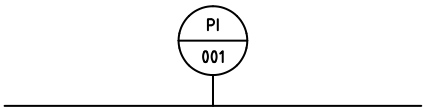
- **Arrow type**– symbol that is inserted in the leader starting point for example an arrow.
- **Arrow size** – determines the leader scale.
- **Insert single segment** –drawing of a one or multi-segment leader.



Procedures

Inserting a marking (multivariant symbol)

1. Run the **Symbols - process control** command.
2. In the **Symbols - process control** select the desired symbol.
3. (Optional) Specify additional markings, numbering method etc.
   For certain fields you can select the value from the drop-down list.
   The ... button will allow you to select the value in an additional window.
4. Specify the rotation **Angle** and **Scale**.
5. (Optional) Enable and change **Leader** settings.
6. Enable/disable the **Insert multiple** option.
7. Click the **Insert symbol** button, to insert the marking into the drawing.
8. Symbol can be used to mark other objects, therefore indicate an object. Depending on the selected marking type, from the selected object user can retrieve data that can be used in the inserted marking.
9. Wskaż punkt wstawienia symbolu oznaczenia.
10. (Optional) Specify the symbol rotation angle.
11. Symbol will be inserted in accordance to the selected settings.
12. (Optional) If in the dialog window the **Insert multiple** option is enabled then repeat the steps 8-11 in order to insert an additional mark or end the command by clicking the **Esc** key.

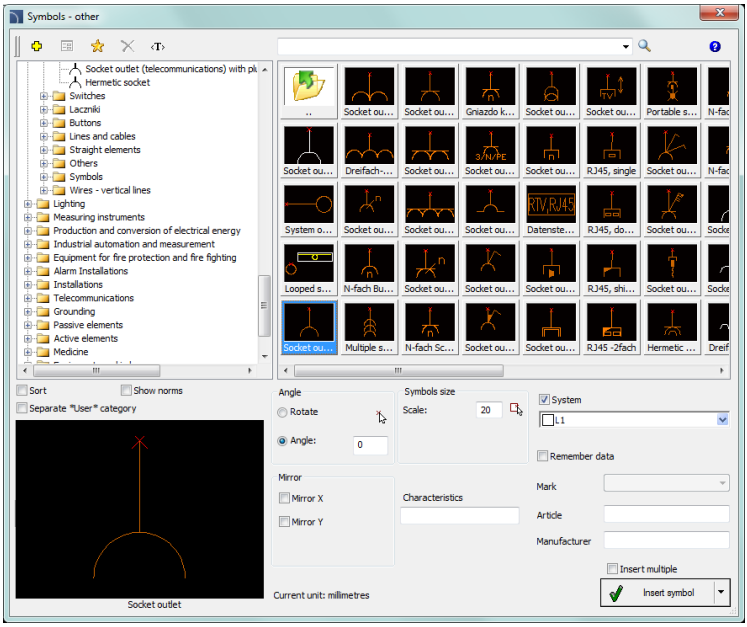


Symbols - other

The **Symbols – other** command contains many symbols that are used in both projections and electrical schemes. Many categories are available, including:

- connecting apparatus,
- symbols used for plans designing (connectors, sockets etc.),
- measuring instruments,
- equipment for fire protection and fire fighting,
- alarm installations,
- telecommunications,
- and many more.

User can add the most frequently used symbols to the ***Favourites*** category thanks to which sure will have a quicker access to them. When selecting symbols, user can also use the find function .



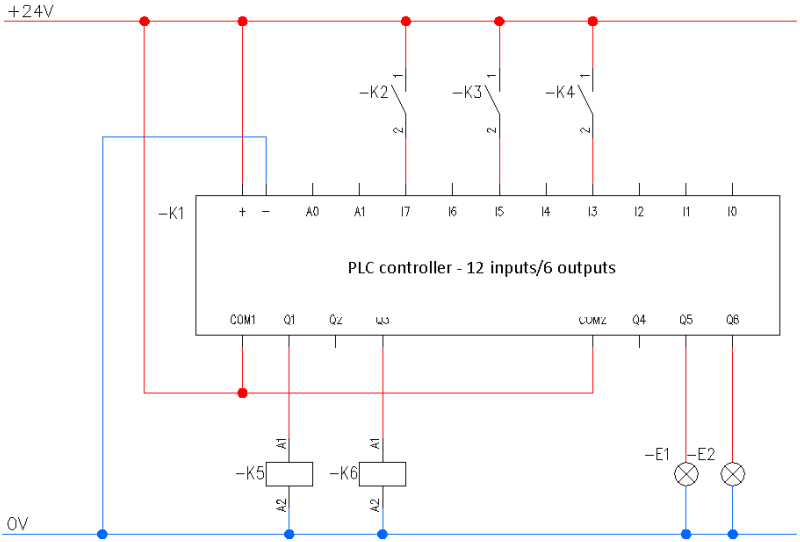
Symbols – other dialog window

Full functionality and all options regarding symbols are described in the **Symbols - IEC, NFPA** (check page 244) chapter.

Sockets and connectors

PLC controllers

The **PLC controllers** command allows to insert to the project any kind of configured blocks of PLC controllers. This command also allows to define any number and type of inputs/outputs, their number and address (numbering).



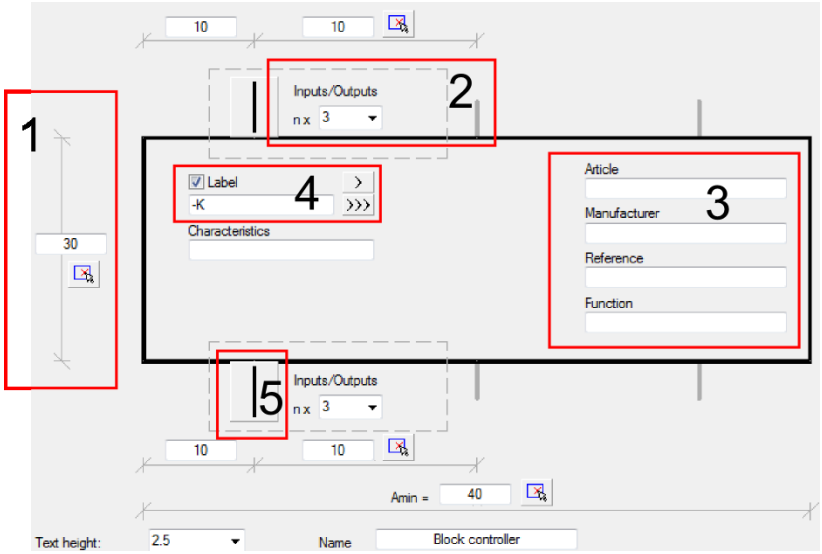
Block of multivariant controllers are symbols for which it is possible to specify all dimensions and customize their appearance and technical parameters. Basic functionality and all options regarding symbols are described in the **Symbols - IEC, NFPA** (check page 244) chapter

When defining a PLC controller block, user should specify its exterior dimensions, and the number of inputs/outputs. After inserting a block, user can describe each inputs/outputs with the help of appropriate signs.
Examples of frequently used controllers configurations, which may be used as a basis in designing user own systems are available in the command.

After inserting the controller into the project it is possible to edit it by using the **Edit symbol** command.

In the dialog window that is used for defining controllers there are active fields and buttons that allow to freely define various parameters.

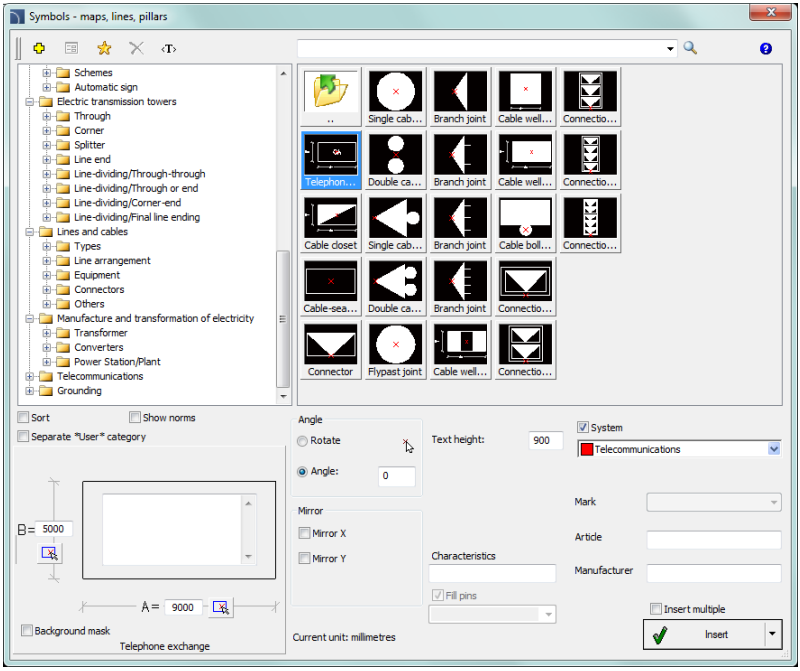
Examples of fields and buttons that are used in the dialog window.



1. **Dimension field** – allow to determine the dimension by entering the value or by indicating two points in the drawing (button).
2. **Drop-down list** – gives the possibility to enter your own value or to select data from the list by pressing the ▼ button.
3. **Text field** – gives the possibility to enter any text.
4. **Label** – gives the possibility to add subsequent labels to inserted symbols. Label field is automatically filled with default marking and the first free number. Thanks to the available buttons, user can choose between the first free number › and a subsequent number ›››. This field can be also filled with any value.
5. **Graphical drop-down list** – expands the pop-up menu and gives the possibility to choose one of available graphical markings.

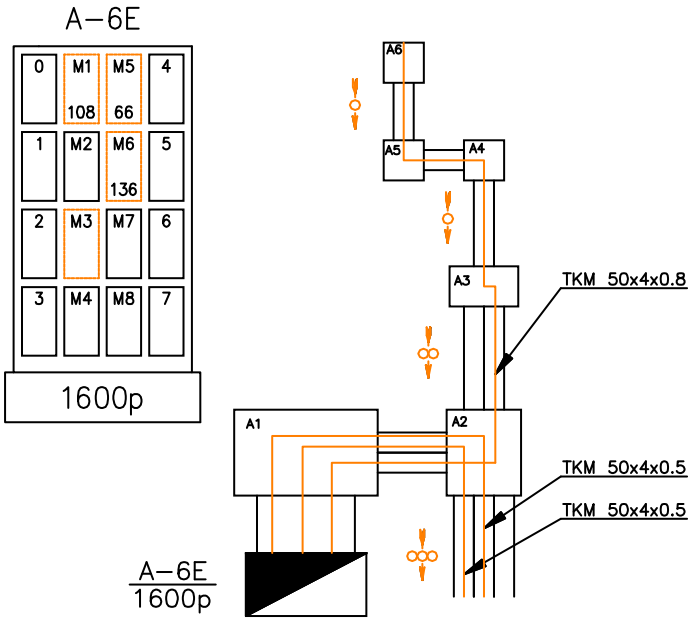
Symbols of maps, lines and pillars

The **Symbols – maps, lines, pillars** command contains a library of symbols, objects and markings that are used in transmission networks and telecommunications drawings.
In the **Telecommunication** category user can find symbols that are used to design copper and fiber networks on maps, as well to create network schemes. Elements that are used in representing the existing, projected and dismantling status are available in the program.
Marking symbols are specially designed for the telecommunication branch and allow to quickly describe objects on maps and schemes.
When designing networks, lines that are in the **Cable, cable trays - schematic** command are also used. Pipe protective casings are available in the **2D Cable trays** command.



Symbols – maps, lines, pillars dialog window

Basic functionality and all options regarding symbols are described in the **Symbols - IEC, NFPA** (check page 244) chapter.



Network scheme example

Example of a network scheme on a map

Descriptions

The **Descriptions** command allows to insert into the project various descriptions that are used in schematic drawings, and in 2D views.

Automatic descriptions

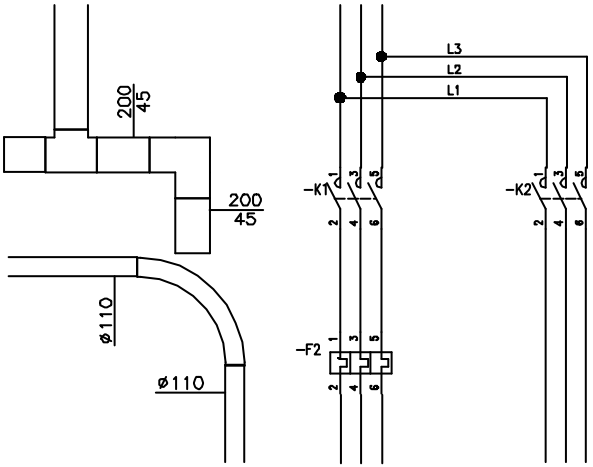
In **CADprofi** program there are commands that are used to describe and number objects. These commands allow to specify technical and identification parameters, which are later used in creating BOMs and gathering various information about the project. Parameters that are added to the objects can be also used in automatic or semi automatic drawing descriptions.

Symbols that retrieve information from objects and insert appropriate descriptions are used for that. When inserting automatic descriptions user has to indicate the described object and next specify the description insertion point.

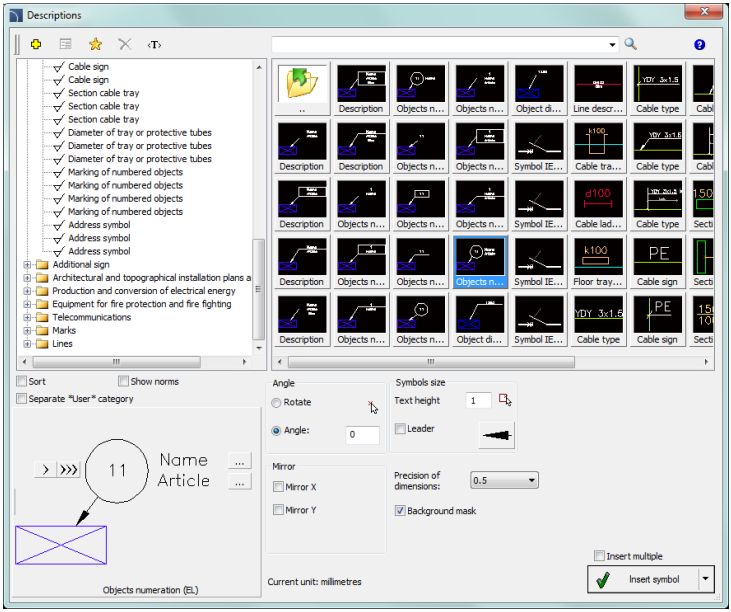
For some symbols it is possible to enable the option to insert them with a leader.

To describe various object user should use appropriate symbols. For example **Line type description** can be used to describe schematical lines. If user uses this symbol to describe other objects, **CADprofi** program won't be able to read the needed data. In this case, the symbol will be described incorrectly.

If it's necessary to describe multiple objects, it's convenient to select the **Insert multiple** option.



Example of automatic descriptions



Descriptions dialog window

Basic functionality and options regarding symbols are described in the **Symbols - IEC, NFPA** (check page 244) chapter. Additional options for the **Description** command:

Text height – specifies the marks text height and scale. User can enter a value or indicate it from the drawing (indicate two points).

Leader – enables/disables usage of leader for descriptions. By clicking on the button, that symbolizes the currently set arrow type, user can change the following leader settings:

- **Arrow type**– symbol that is inserted in the leader starting point for example an arrow.
- **Arrow size** – determines the leader scale.
- **Insert single segment** –drawing of a one or multi-segment leader.

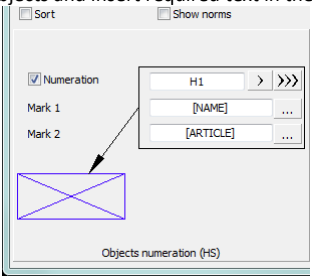
Precision of dimensions – defines the precision (decimal number) of all values of dimensions that are placed in marks. These dimensions are automatically retrieved from the indicated object.

Background mask – enables/disables the possibility to mask the symbol background by adding a WIPEOUT. This feature is helpful if user needs to highlight important information in the drawing.

Multivariant marks and object numbering

Many mark symbols are multivariant symbols, for which user has got a possibility to customize the appearance and to select additional options. For these symbols in the dialog window there are selection fields and text fields buttons available that allow to specify the variant or adoption of symbol to current needs.

An example of such multivariant symbol is the "Objects numeration" symbol. This symbol can number objects and also retrieve from them required parameters (e.g. name, type etc.). In this way it is possible to quickly number many objects and insert required text in the drawing.

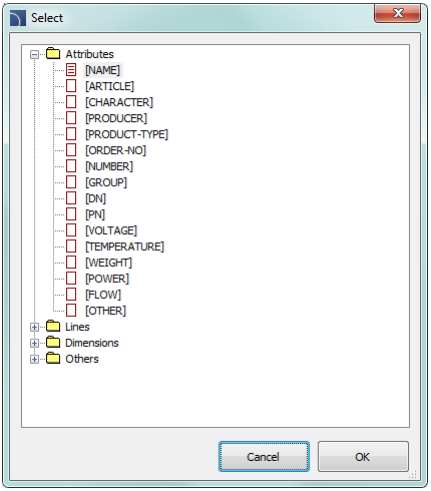


CADprofi program automatically detects numbered objects in the drawing, thanks to which in the dialog window the first available number from the specific branch (e.g. EL - electrical installations) is being displayed. The button allows to select the next highest number.

By adding different number prefixes, user can create many numbered sequences (e.g. one F1, F2, F3 sequence for switches and one K1, K2, K3 for relays).

Inserting symbols with the **Insert mupltiple** option will cause the symbols to have numbers added to them with increased numeration.

The **Mark 1**, **Mark 2** fields allow to add in the selected symbol any text or information retrieved from the indicated object. Selecting the information type will be possible after pressing the button that is located next to the field.



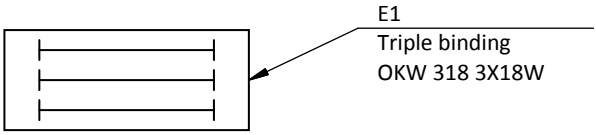
In the newly opened window user can select the following information:

- **Attributes** – indicated object attributes values.
- **Lines** – line type or installation kind.
- **Dimensions** – dimensions retrieved from the indicated object.
- **Others** – for example the CAD program system variable values.

Procedures

Inserting an automatic description (multivariant symbol)

1. Run the **Descriptions** command.
2. In the dialog window select the appropriate description symbol.
3. (Optional) Change the data type by clicking the **Browse** button and select the data from the list of available data in the **Select** dialog window.
4. Specify the parameters: **Angle**, **Mirror** and **Scale**.
5. (Optional) Enable and change **Leader** settings.
6. Enable/disable the **background mask**.
7. Enable/disable the **Insert multiple** option.
8. Click the **Insert symbol** button, to insert the desired mark into the drawing.
9. Indicate the symbol/object whose data should be inserted into the drawing.
10. Indicate the mark insertion point.
11. (Optional) If the **Insert single segment** option (in the leader settings) is turned on then indicate a series of points that are segments ends or click the right mouse button to continue.
12. (Optional) Specify the mark rotation angle.
13. (Optional) If in the dialog window the **Insert mulptiple** option is enabled then repeat the steps 10-13 in order to insert an additional mark or end the command by clicking the **Esc** key.

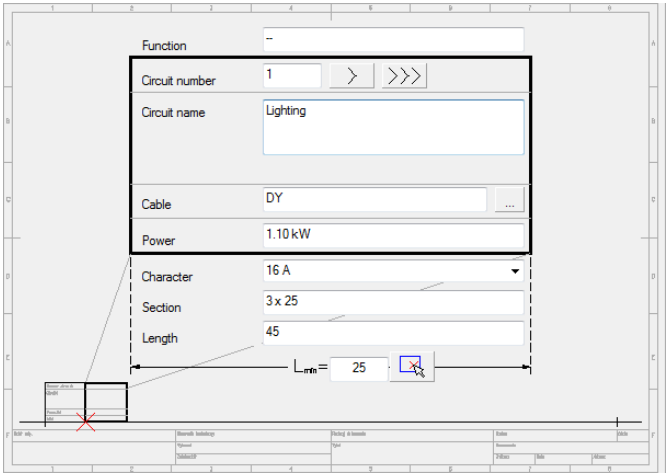


An example of a description (marking) with number, object name (Name attribute) and type (Article attribute).

Frames and tables

Frames and drawing tables are available in **CADprofi Electrical**. Various drawing tables are also available in general **CADprofi** commands, but when user creates electrical projects, it is recommended to use the tables from the **Electrical** module, which are specially prepared to electrical schemes.
More information about frames and tables can be found in the **Tables** (check page 37) chapter.

Circuit markings:
Frames and tables command allows to insert tables with circuit markings. These tables are being inserted in the lower part (most often directly under the circuit scheme). Several table arrangements are available. In the first step user should insert to the project the table header and afterwards the tables that contain data for each circuit. In the circuit tables it is possible to provide the circuit name and number, as well as additional parameters, such as: power, type, security type, kind and line length.

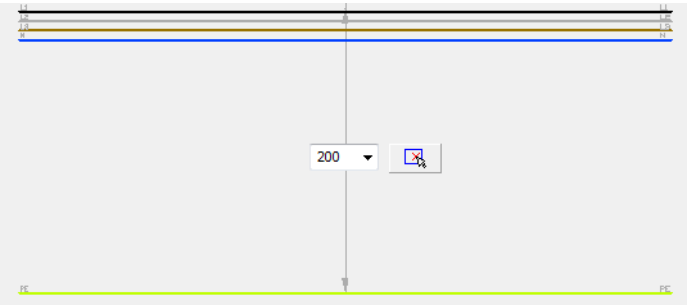


Frames and tables dialog window

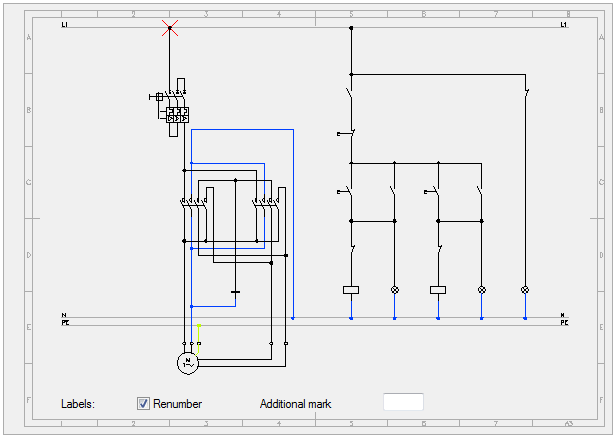
> ***Hint:***
>
> ***Data from circuit tables can be used in BOMs. In the BOM creator user should select the „Electrical – summary based on tables circuit" (check page 46).***

Scheme templates

The **Scheme templates** command allows to insert to the project predefined lines potentials and circuit lines, which are used to quickly create schemes.
Predefined lines are adapted to suit both **ISO** and **ANSI** standards.
Lines potentials are parametrical, thanks to which it is possible to specify spacing between the scheme lines.

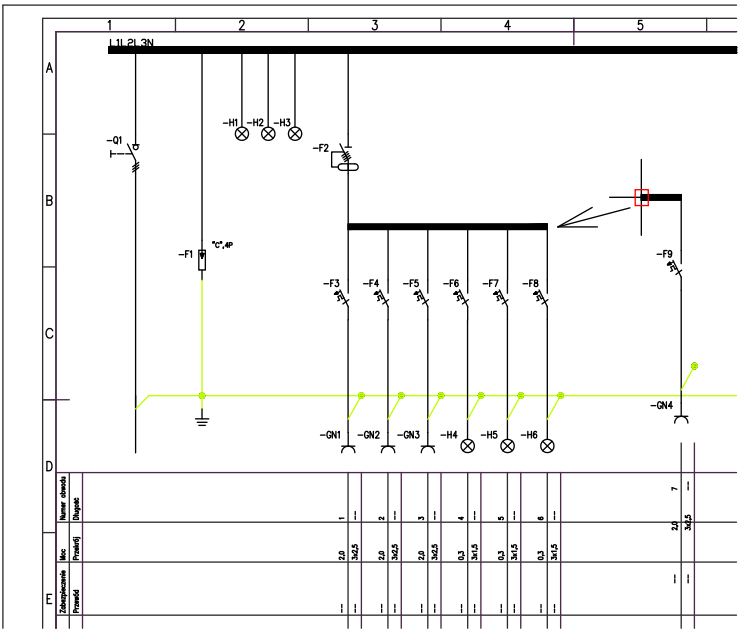


The **Scheme templates** command contains predefined electrical schemes that can be used to create typical schemes. Direct-on-line start of 3-phase motors, reversing systems, star-delta starting and many more schemes are available.
Each scheme has got components that are initially pre-numbered. User can enable **renumbering** and specify **additional marks**.



A special group of schemes are distribution centre - circuits. Sets that contain several circuits are available, as well as single lighting or socket circuits. Thanks to this user can instantly

create various distribution centre schemes. Socket and lighting circuits can be used for other types of circuits - after inserting such a circuit, all that user has to do is to change the description and the receiver symbol.
When inserting distribution centres, tables that are used to specify the name, number and circuit parameters are also inserted. To edit this data, user can use the **Edit symbols** command or other commands that are used to edit block attributes.



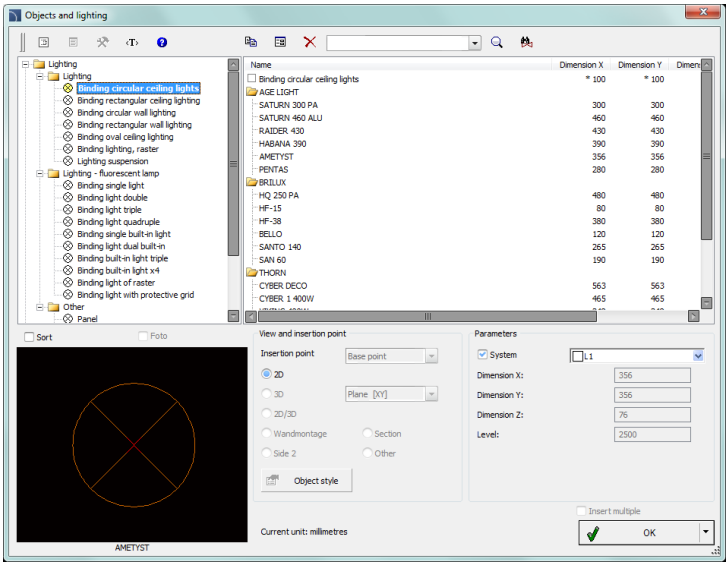
Example of a typical scheme with insertion of an additional circuit

Schemes: sheet templates

Lighting

The **Objects and lighting** command allows to insert typical lighting fixtures. This command also contains sample light fixtures that were prepared on the basis of certain manufacturers catalogs. Before using these lighting fixtures in projects it is advised to check the current offer of these manufacturers.

Lighting fixture is also available in the **Symbols - other** command. However, it should be noted that symbols are elements that contain only a representation of a specific device and therefore do not need to have the actual dimensions. The **Objects and lighting** command on the other hand gives the possibility to insert object to the project, whose dimensions match the dimensions of actual devices.



Objects and lighting dialog window

Basic functionality and options regarding symbols are described in the **Symbols - IEC, NFPA** (check page 244) chapter. Additional options for the **Objects and lighting** command:

Parameters – first element in each category gives the possibility to determine the dimensions. After selecting a such element, user has to specify dimensions in the Dimension X, Y, Z fields.

View and insertion point – for some objects additional views are be available, e.g. side views that allow to insert objects not only on plans but also on cross-sections and other installation views.

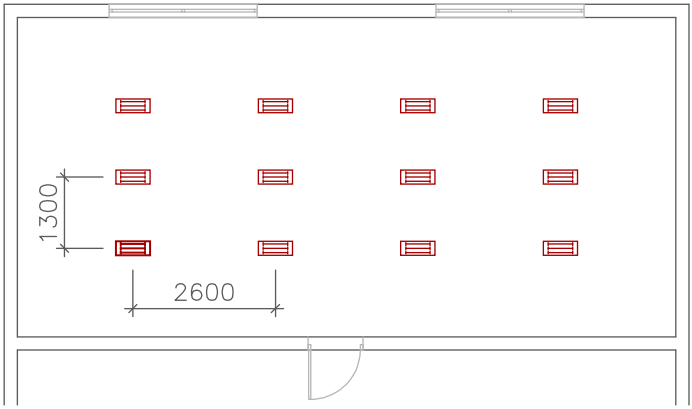
Procedures

Inserting a lighting fixture

1. Run the **Objects and lighting** command.
2. In the **Objects and lighting** dialog window expand the needed category.
3. In the detailed list that contains lighting fixture types a list of all available manufacturers or series of types is available.
4. (Optional) Double click on a series of types in order to expand it and display its content.
5. Select an element by clicking a position from the list. If the first element will be chosen, then it will be possible to specify the X, Y and Z dimensions, which allows to insert to the project lighting fixture with non standard dimensions.
6. (Optional) Change the **View** for the selected element.
7. Disable the **System** option or select an installation from the available list, to specify the layer for the selected element.
8. Click the **OK** button to insert the selected object to the drawing.
9. Specify the insertion point and the object rotation angle.

Lighting

> ***Hint:***
>
> ***If the lighting fixture should be located on a net then after inserting the first lighting fixture user can use the ARRAY (_ARRAY) command from the CAD program.***

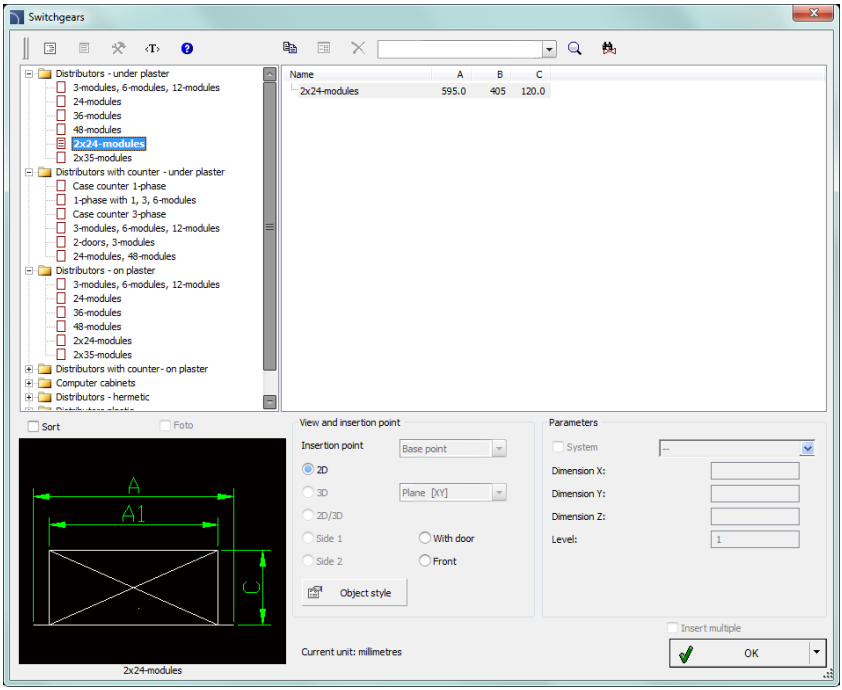


Example of inserting a lighting fixture: *After inserting the first lighting fixture, the rest fixtures were copied by using the ARRAY command with the distance 2600 mm x 1300 mm.*

Switchgears

The **Switchgears** command gives the possibility to insert switchgear (distribution). **2D** views (from the top) are used when designing on plans. Front views are used when designing distribution facade.

This command allows to select the **Insert (specify all dimensions)** option, thanks to which user can insert objects with custom dimensions. It is also possible to save the user dimensions by Creating and extending the user type of series **Creating and extending user type of series** (check page 88).



Switchgears dialog window

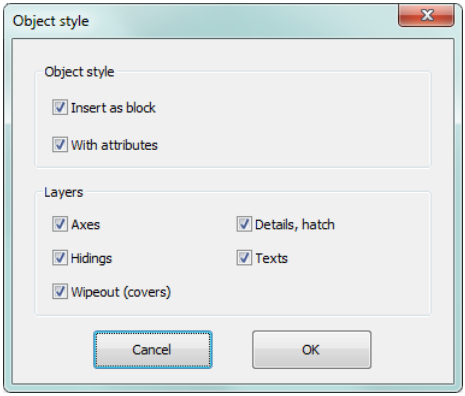
The method of working with object has been described in previous chapters.

This command also contains the following items:

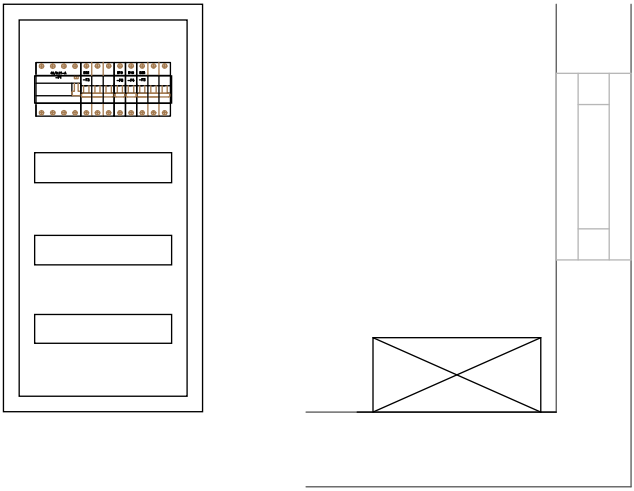
Object style – options that allow to specify the object style and the level of object detail.

- **Insert as block** –allows user to insert an element that is exploded into component parts or a one single object (block).
- **With attributes** – enables/disables adding of attributes to the inserted object.
- **Axes** – enables/disables an option to draw axis in the inserted object.

- **Hidings** – enables/disables drawing elements that are hidden in an object.
- **Wipeout (covers)** – enables/disables addition to the inserted elements **Covers** (WIPEOUT) type objects.
- **Details, hatch** – enables/disables drawing of hatches and object details such as holes in the flanges, arrows that indicate the direction flow and others.
- **Texts**– enables/disables drawing of potential texts in objects.



Object style dialog window



An example of switchgear on a plan and a facade with inserted apparatus

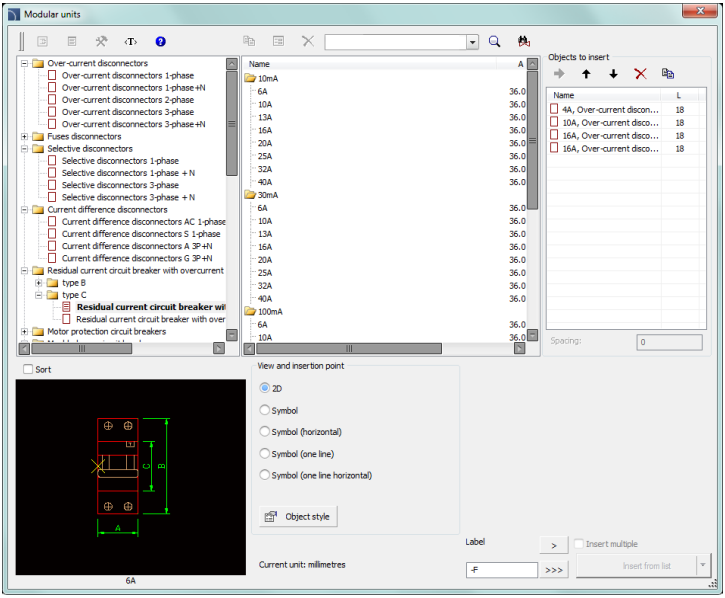
Designing switchgear

Modular units (apparatus)

The **Modular units** command contains many types of connection apparatus, protection and other components used in electrical projects. For all apparatus both views used when designing switchgears and when symbols on schemes are applied. In case of inserting apparatus views it is possible to create a **list of objects to insert**, which is also used to simultaneously insert many objects to the drawing.

If user inserts symbols he should use all policies and procedures described in the **Symbols - IEC, NFPA** (check page 244) chapter.

After inserting apparatus view it is possible to use the **Create alternative view** command in order to obtain in a convenient way symbols for each apparatus. It is also possible to obtain apparatus views on the basis of previously inserted symbols. Using this command user can quickly create a scheme based on the switchgear view or vice versa (check page 64).



Modular units dialog window

The method of working with object has been described in previous chapters.

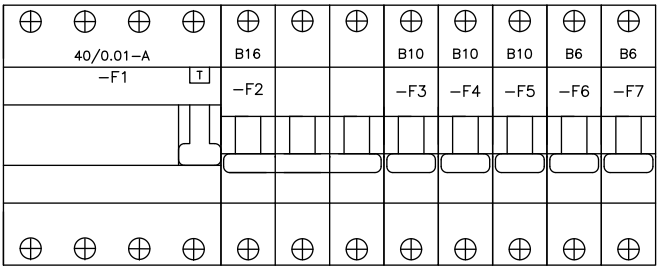
This command also contains the following items:

Objects to insert – list of objects that will be inserted into the drawing.

The **Add element to a list** button adds object to the list of objects to insert.

Insertion options – when inserting a single, user can use other option than the default one to insert an object in the drawing. The button expands the list of available insertion options:

- **Insert (specify style for object)** – inserts selected object with a possibility to change options in the **Object style** dialog window.
- **Insert (specify all dimensions)** – inserts selected object with possibility to specify all dimensions that can be edited. It is also possible to save the user dimensions by Creating and extending the user type of series **Creating and extending user type of series** (check page 88).



Procedures

Inserting many objects (list of objects to insert)

1. Run the **Modular units** command.
2. In the dialog window select the appropriate category.
3. In the detail a list of available apparatus types will be visible.
4. (Optional) Double click on a series of types in order to expand it display its content. In order to expand all series of types click the right mouse button and from the context menu select the appropriate option.
5. (Optional Change the **View** for the selected element.
6. (Optional) Change the **Object style** settings.
7. Select an element by double clicking or select it and click the **Add element to a list** button.
8. Continue with filling the list by repeating the steps **2 - 7**.
9. (Optional) Organize the list with the help of the **Copy**, **Delete** **Move selected rows down** or **Move selected rows down** buttons.
10. Specify the label and number of the first apparatus. Next apparatus will be inserted with consecutive numbers.
11. Click the **OK** or **Insert from list** button, to insert object into the drawing.
12. Specify the insertion point and the objects rotation angle.

Designing switchgear

Automatic obtaining of apparatus view upon symbols

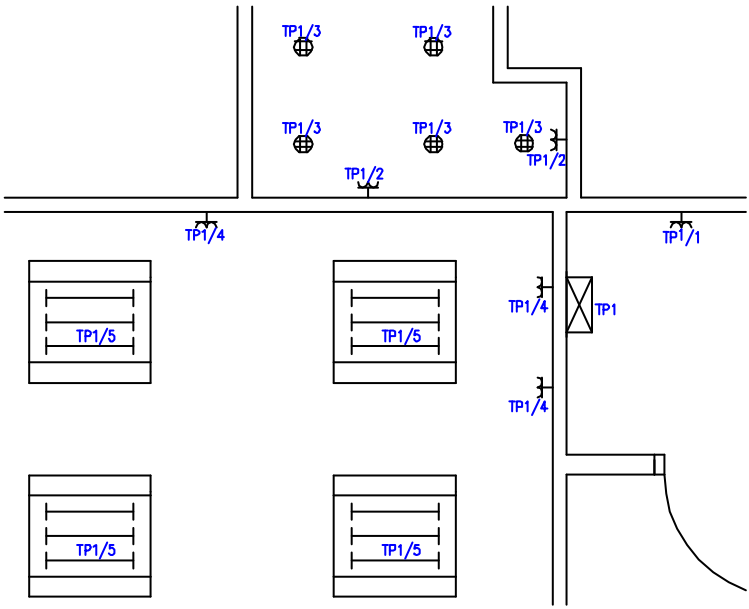
Circuits numbering

The **Circuits numbering** command allows to define electrical circuits and assign them to indicated objects (addressing objects).
CADprofi Bill of materials creator allows to prepare specifications that include the assignment of objects to individual circuits.

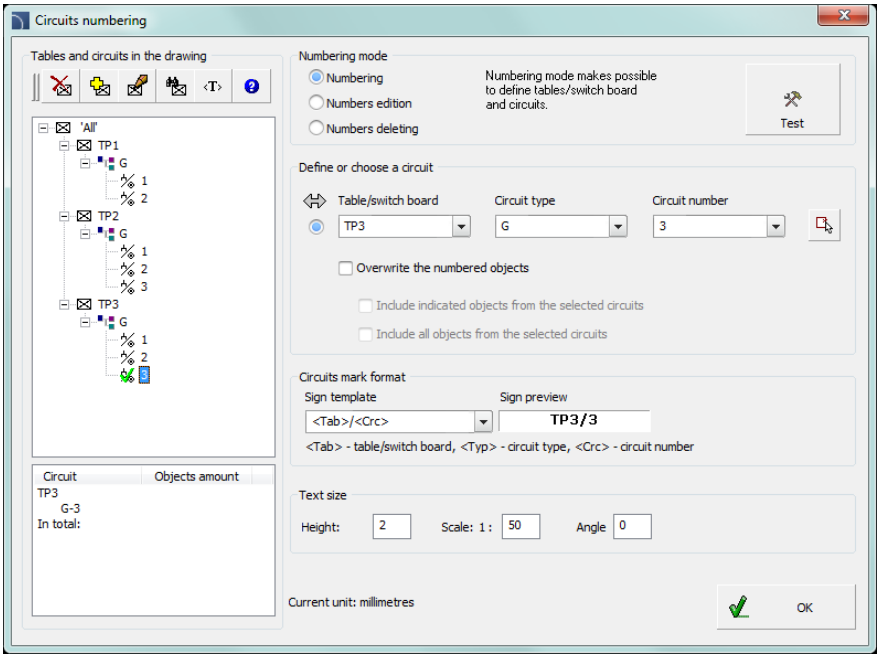
This command can work in three modes:

- **Numbering** – this mode is used to define circuits and address objects. Each object receives a circuit number to which it will be connected.
- **Numbers edition** – this mode allows to edit circuit numbers (addresses) for many object simultaneously.
- **Numbers deleting** – deletes numbers from selected objects.

During the defining of schemes program allows to indicate switchgear on the drawing, which can be any object or symbol from the **CADprofi Electrical** module, e.g. an element inserted with the **Switchgear** command. It is possible to define circuits, for which there is no switchgear in the current drawing.



Example of objects with defined circuit numbers



Circuit numbering dialog window

The **Circuit numbering** dialog window contains the following elements:

Tables and circuits in the drawing – list of tables and their circuits displayed in a form of a drop-down tree. Symbols on the list:

- **Table/switch board** –
- **Circuit type** –
- **Circuit number** –

In the bottom part of the window the information about number of objects in each circuit is being displayed. If the main element of the tree is being selected, then a detailed list of all circuits and objects will be visible.

Menu list of tables and switch boards – tools that are used to manage boards (switchgear).

Delete number from the table – deletes number from the block that symbolizes the indicated board.

Indicate a table on the drawing – allows to indicate (or change) in the drawing the block which symbolizes the indicated board.

Change the table number – allows to change the board name.

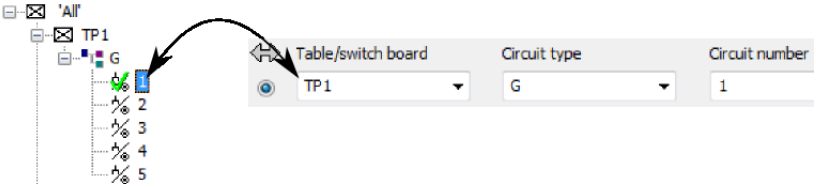
Zoom on a circuit or a board – this command selects objects that are assigned to the indicated circuit or circuits. If switchgear will be chosen, then the zoom is performed on a drawing object (block), which acts as a board.

Test/Repair – this option allows to automatically detect and correct any possible inconsistencies in the current circuit numbering.
It is recommended to execute this command after copying, deleting and moving objects in order to synchronize changes with the inserted numbers.

Define or choose a circuit – a group of options that allow to define new circuits or operations (edit and delete) the existing circuits.

- **Table/switch board** – specifies the table marking.
- **Circuit type** – circuit type marking.
- **Circuit number** – circuit number.

Information about table, type and circuit number can be filled by clicking the needed position on the **list of tables and circuits**.



There is a possibility to fill above data by indicating an object on the drawing, which was already assigned to a circuit. In this case user should click the **Take circuit number from the object in the drawing**.
If the **edition** mode is enabled, then in the **Circuit defining or choosing** additional **Change for** field will be available.

Overwrite the numbered objects – enable/disable the possibility of overwriting an existing numbering. This option is only available in the **Numbering** mode.

Options used for object editing/deleting:

- **Include indicated objects from the selected circuits** – gives the possibility to edit/delete addressing from selected objects.
- **Include all objects from the selected circuits** – edits/deletes all objects that belong to a specified circuit.
- **Delete numbering only from selected circuits** – when this option is enabled it is possible to delete addressing only from objects that belong to a specified circuit.

Circuits sign format – this option allows to determine the way how to show in the drawing the information about circuit numbering. Sign template can contain any text and

circuits data (**<Tab>** – table/switch board indication, **<Type>** – circuit type, **<Crc>** – circuit number). During the template definition in the next field a preview is being displayed, which will be inserted in the drawing.

Text size – specifies the circuit sign angle rotation and size.

- **Height** – specifies the text size on printings.
- **Scale** – specifies the text scale on printings.
- **Angle** – specifies the circuit sign rotation angle.

Procedures

Object numbering

1. Run the **Circuits numbering** command.
2. In the dialog window in the select the **Numbering Mode**.
3. In the **Define or choose a circuit** specify the table name, circuit type and number or if it is possible indicate from the list a circuit to which you would like to add objects.
4. (Optional) Enable/disable the **Overwrite the numbering objects** option.
5. Specify the circuit **sign format**.
6. Specify the **Text size** options.
7. Click the **OK** button. Window will be closed and it will allow to select objects that will be assigned to a specific circuit.
8. (Optional) If in the table/switch board a new name was entered then the application will propose to indicate the table on the drawing.
9. Indicate the first element that will be added to the circuit.
10. Specify the placement of the numbering marking text.
11. (Optional) Indicate objects that you would like to add to the selected circuit. Confirm by pressing right mouse button or the **Enter** key. Numbering markings will be automatically placed according with the first marking spacing.

Numbering edition

1. Run the **Circuits numbering** command.
2. In the dialog window in the select the **Numbers Edition Mode**.
3. In the **Define or choose a circuit** group select the table name and a circuit type and number, from which objects you would like to edit.
4. In the **Change for** fields the table name and circuit type and number, for which you would like to assign selected objects. Entering a new table name, type or circuit number will cause a creation of a new circuit and assignation of objects to it.
5. (Optional) Enable/disable the **Include indicated objects from the selected circuits** option.
6. (Optional) Enable/disable the **Include all objects from the selected circuits** option.

7. (Optional) Specify the circuit **sign format**.
8. (Optional) Specify the **Text size**.
9. Click the **OK** button.
10. (Optional) If in the table/switch board a new name was entered then the application will propose to indicate the table on the drawing.
11. (Optional) If the **Include indicated objects from the selected circuits** option was enabled then indicate elements, which numbering you would like to change. Confirm by pressing right mouse button or the **Enter** key.
12. Selected elements will be transferred to the indicated circuit, and their marking will be changed.

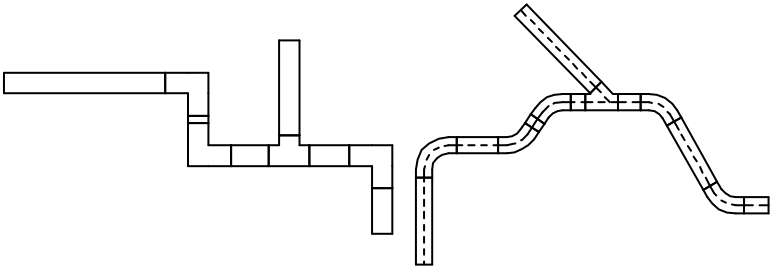
Deleting numbering

1. Run the **Circuits numbering** command.
2. In the dialog window in the select the **Numbers deleting Mode**.
3. (Optional) Enable the **Delete numbering only from selected circuits** option.
4. (Optional) Enable the **Include indicated objects from the selected circuits** or **Include all objects from the selected circuits** option.
5. (Optional) In the **Define or choose a circuit** card select the table name and circuit type and number, from which you would like to delete numbering.
6. (Optional) Enable/disable the **Include specified objects from various circuits** or **Include all numbered objects from a drawing!!!**
7. Click the **OK** button.
8. (Optional) If the **Include indicated objects from the selected circuits** or **Include specified objects from various circuits** option was enabled then indicate elements, which numbering you would like to delete. Confirm by pressing right mouse button or the **Enter** key.
9. Numbering of selected elements will be deleted.

Circuit numbering (addressing)

2D Cable trays and ducts

In **CADprofi** program it is possible to create plans and installations cross-sections that include trays, ducts and protective tubes. When drawing an installation line elements and fittings are available (elbows, reducers, tees etc.). Each fitting is a parametrical object, which is being drawn with appropriate program's procedures. Thanks to this it is possible to draw any fittings with specified dimensions. For each element, databases are available, that contain definitions of elements of selected manufacturers.



Tray *Protective tube*

Trays and fittings are available in the following commands:

- **2D Cable trays** – this command contains typical trays and fittings used when designing cable trays.
- **2D Cable trays - sequence** – this command allows to draw many segments with automatic insertion of elbows.

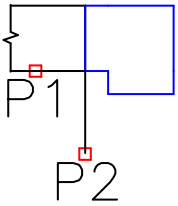
Trays and ducts

The **2D Cable trays** command allows to insert single trays or ducts to the drawing.
After running the command user has to select the needed fitting or tray, specify its dimensions and insertion options. Program possesses functions that automate the connection of elements to each other. It is possible to automatically adapt the tray size and connect it to other elements. Below there are examples with usage of this function.

Examples of inserting fittings

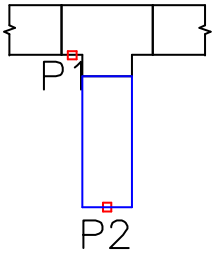
Automatic insertion at the end of the tray.

1. Indicate a tray to which you would like to insert an elbow (**P1**).
2. Specify the elbow direction (**P2**).



Inserting a tray with the „join object" option.

1. Indicate and object to which you would like to connect a tray (**P1**).
2. Specify the length of a new tray (**P2**).

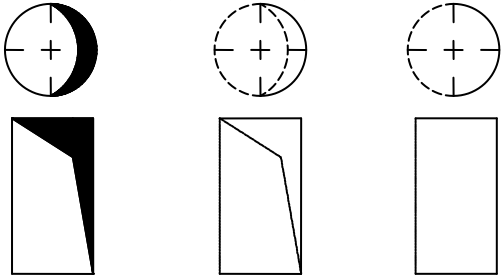


When connecting trays or ducts to other installation elements the **Diameter fit** option is handful, cause thanks to it the inserted element always has the same size, so it will suit the indicated object.

Z+/Z– trays and fittings

Z+/Z- trays represent the vertical tray segments (along the Z axis). On plans for such elements the cross section symbol is also visible. Because of that Z+/Z- tray length has to be specified in the dialog window before adding them to the drawing.

For Z+/Z- trays and fittings it is possible to additionally specify the symbol cross section (Z+, Z- or without symbol) and the insertion point (beginning, axis, end).

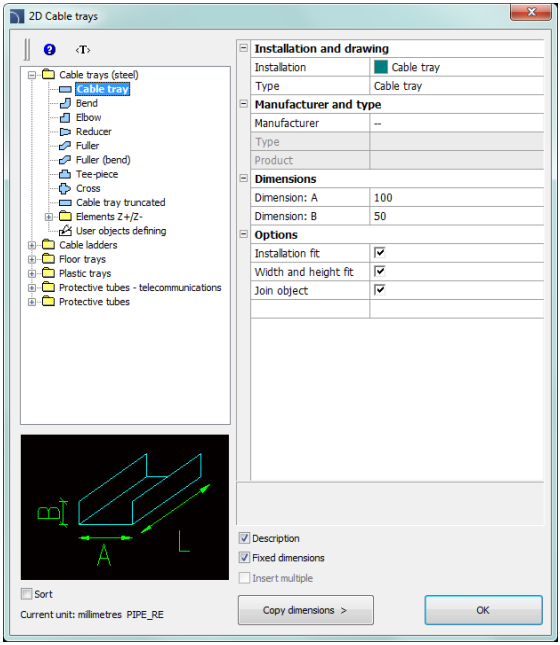


Z+ *Z–* *Without any symbol*

Retrieving fittings dimensions

During drawing it is often needed to insert similar elements to elements that are already inserted in the drawing. In such case it is convenient to retrieve dimensions from another fitting. After pressing the **Copy dimensions** button user has to indicate the needed fitting. It is important to avoid indicating fittings that have completely different parameters than the element currently chosen in the dialog window. In such cases, user may experience ambiguity in the data interpretation.

For example: if an elbow was selected in the dialog window, and user selects a reducer then **CADprofi** won't be able to specify the elbow angle and the radius, because reducer doesn't posses such parameters.



2D Cable trays dialog window

The **2D Cable trays** dialog window contains the following items:

Categories tree – a set of all categories and symbols in the library, shown as a pull-down tree.

Preview – displays the preview of the currently selected element. Right-clicking on the preview window will zoom the view.

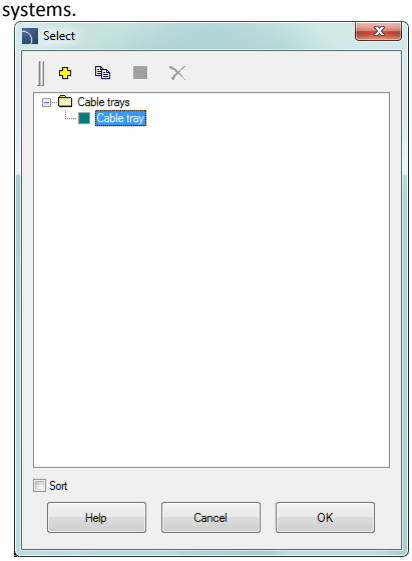
Sort – enables/disables categories alphabetical order.

Fixed dimensions – enables/disables the preserve of a basic dimension after inserting objects such reducers. After inserting such element in which the line dimension is being reduced in the next time user runs the command the application will automatically change the default diameter into a reduced diameter (**D2**). This process is very convenient when designing an installation sequence. However, if user wants to insert a fitting that has an unchanged main cross-section then enabling the **Fixed dimensions** option will prevent an automatic change of cross-section.

Installation and drawing:

- **Installation** – gives the possibility to choose the installation kind or define a new one. Initially on the drop-down list only the most important **CADprofi** systems will be included. However, user has the possibility to develop the list with his own elements (by using the **Add to most recently used** option), which is located in the

Lines, cable trays - scheme (check page 222) dialog window. Selecting the **Others/Edit** option will open a new dialog window with full list of available systems.



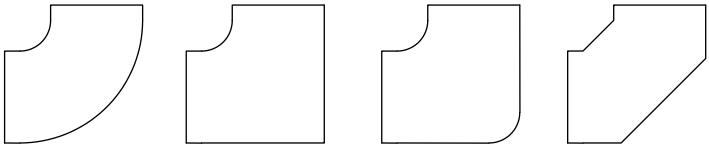
Define new – allows to add a new position (tray) to the list as a user tray. Afterwards it opens a new **edit** window and allows user to edit the line data.

Copy – copies the selected line with all its parameters, therefore creating an element made by user.

Edit – opens the edit properties window of user created lines.

Delete – deletes the selected line from the list.

Type – possibility to specify the fitting type (normal element, segment etc.).

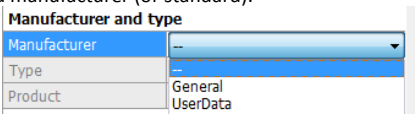


Elbow types available in the program

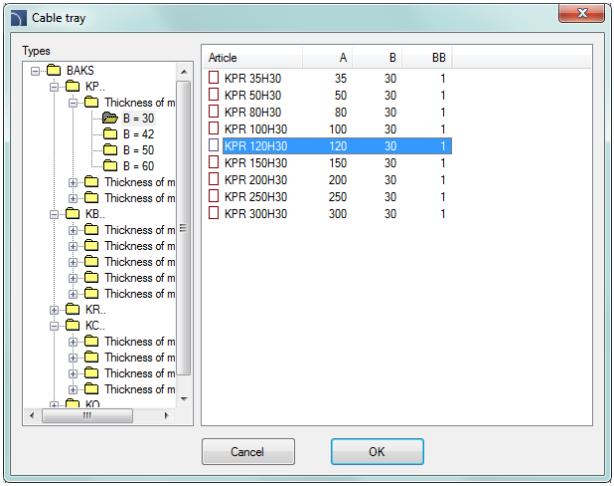
Manufacturer and type – selection of items from manufacturers' standards or databases.

CADprofi program allows to draw fittings and 2D trays, whose dimensions can be specified in two ways:

1. Drawing based on dimensions specified by user.
   To enable this mode, in the **Manufacturer** field select „**– –**".
2. Drawing based on manufacturers' standards or databases. In order to activate this mode select the required manufacturer (or standard).
   After selecting the manufacturer it is not possible to enter you own dimensions (this option is blocked).



After selecting a specific manufacturer or standard a dialog window will appear with available fittings or lines.



Element selection dialog window

Dimensions – In **CADprofi** program it is possible to draw fittings with any dimensions. To do this, in the **Manufacturer** field select „**– –**"
When drawing is based on user selected dimensions then two field in which user can enter values are available. button allows to select typical values. Trays length is determined in the drawing by indicating two points.

Options – additional options that automates the drawing process:

- **Join object** – automatic connection of inserted element to existing objects. When connecting, **CADprofi** program will automatically determine the rotation angle and the element insertion point.
- **Installation fit** – automatic determination of installation kind based on object, to which the inserted element is being connected. Installation kind set in the dialog window is being ignored.
- **Diameter fit** – automatic identification of installation diameter based on object to which an element is being connected. In the connection point it is possible to insert a reducer or to ignore the line diameter from the dialog window.
- **Insert in a sequence** – gives the possibility to insert fittings into an existing line or into their extension. If this option is disabled then fittings are inserted on the end.

Procedures

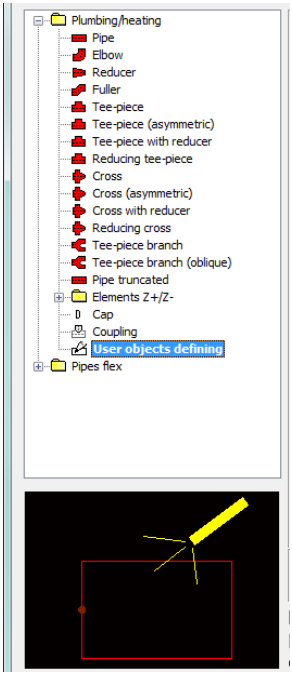
Inserting a tray

1. Run the **2D Cable trays** command.
2. In the **2D Cable trays** dialog window from the list of available categories select **Tray**, **Duct** or **Protective tube**.
3. From the **list of installations** select the needed kind of installation in order to specify the color and layer for a new tray.
4. (Optional) Specify the values for the **Manufacturer and type** items.
5. In the **Dimensions** field specify the **Diameter** value for the tray.
6. (Optional) Click the **Copy dimensions** button and indicate an element on the drawing from which you would like to copy the diameter value.
7. (Optional) Enable/disable the **Installation fit**, **Diameter fit**, and **Join object.**
8. Click the **OK** button to start drawing a tray.

Cable trays and ducts

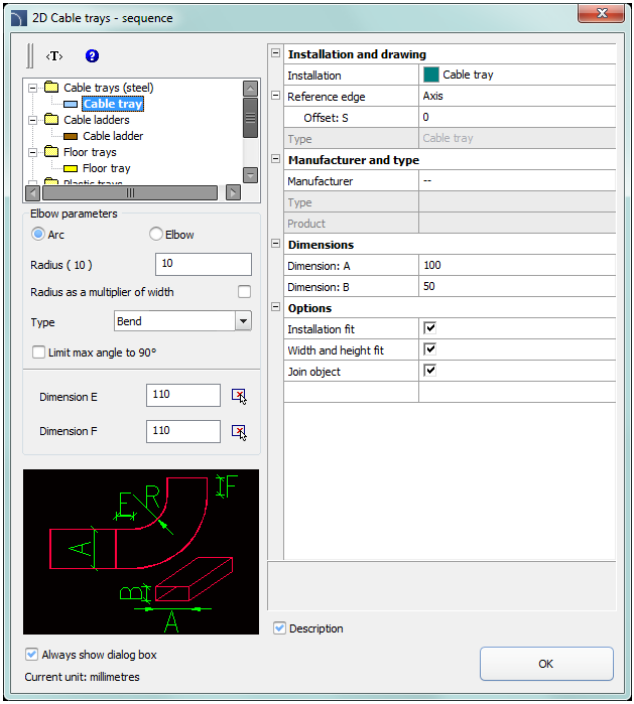
Defining user own fittings

When designing installations it is sometimes needed to use non-standard fittings. In this case, user can define his own fittings, which will be taken into account during numbering or during creating BOM.
In order to define a new additional fitting, user should draw it first and then run the **2D Cable trays** command. After that user should choose the User object defining option. For the newly created fitting user should choose a name. Optionally user can also specify the manufacturer, type and product parameters.
After clicking the **OK** button, user should select objects from the list that the new fitting is being made up from and to choose the base point.
If a fitting is often used then it is possible to add it to the program library by using the **User block wizard** (check page 87).



Drawing tray sequences

The **2D Cable trays - sequence** command allows to create trays and protective tubes sequences with automatic insertion of elbows that have parameters defined by user (or without elbows). This command has the ability to disable the display of the dialog box, allowing user to immediately start drawing trays sequences with the last used settings.



2D Cable trays - sequence dialog window

Graphical layout and basic functionality of the dialog window is similar to the **2D Cable trays** (check page 276). Additional options available in this command:

Elbow parameters – a group of options that allow to specify parameters of elbows.

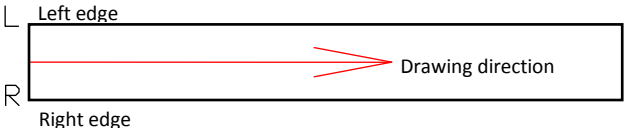
- **Radius** – allows user to determine the radius or radius ratio for elbows inserted between each tray segments.
- **Radius as a multiplier of width** – enables/disables the possibility to determine the radius through the width multiplier.
- **Type** – allows user to choose the elbow type that will be used in the sequence.

- **Limit max angle to 90°** – enables/disables the possibility to reduce the line sequence elbow angle to 90°.

Dimension E/Dimension F – allows to specify both E and F dimensions (that are visible on the preview). button allows to specify the dimension by indicating two objects on the drawing.

> *Hint*
>
> *If the „Always show dialog box" option was disabled then if user wants to enable it again he has to choose Settings (abbreviation „S" in the command line) after running the command.*

Reference edge – right or left edge. Edge is indicated accordingly to the drawing direction.



Procedures

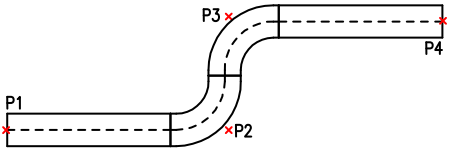
Inserting trays

1. Run the **2D Cable tray – sequence** command.
2. (Optional) If the dialog window didn't appear then start drawing or select the **Settings** (S key) option, to change the settings.
3. In the dialog window choose the needed kind of tray.
4. Choose installation kind.
5. (Optional) Specify the values for **Manufacturer and type** controls.
6. Specify the line **Diameter** (or cross-section) for the tray or protective tube.
7. (Optional) Enable/disable the **Installation fit**, **Diameter fit**, and **Join object.**
8. Click the **OK** button to start drawing a tray sequence by indicating the points.

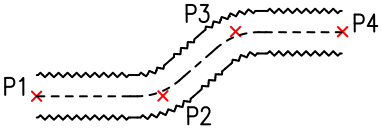
Cable trays and ducts

How to draw

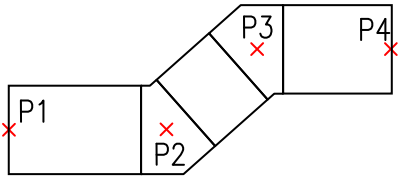
When drawing users should indicate points (**P1**, **P2** etc.). If you draw by the axis and without an offset line segments are being inserted along with the indicated points.



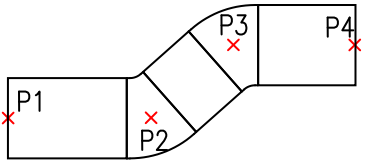
Protective tube



Flex tube



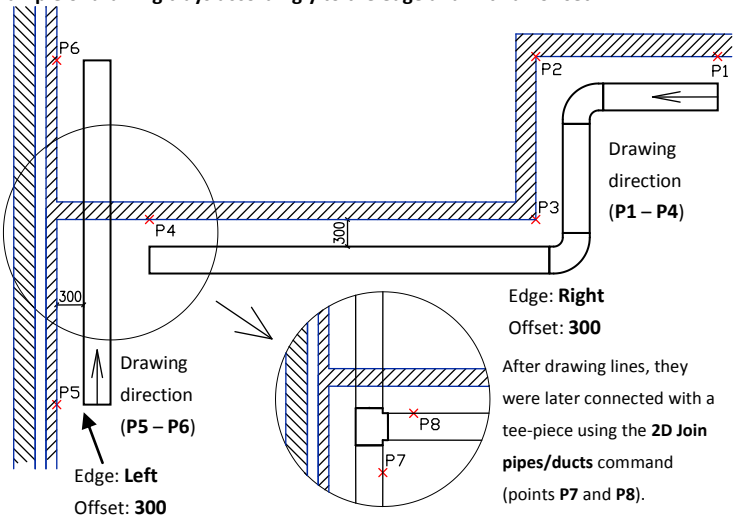
Rectangular pipes with elbows



Rectangular pipes with arches

When drawing with an offset the tray sequence is being drawn with a specified distance from the indicated points.

An example of drawing trays accordingly to the edge and with an offset:

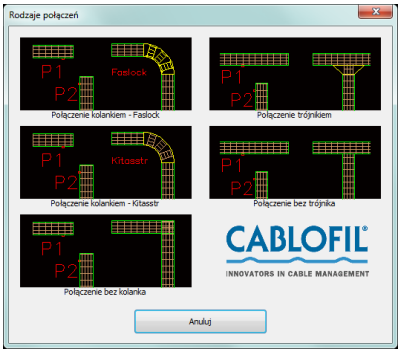


After drawing lines, they were later connected with a tee-piece using the **2D Join pipes/ducts** command (points **P7** and **P8**).

Drawing mesh trays

In **CADprofi** program it is possible to draw **Cablofil** mesh trays.
This system is available in the **Manufacturers library** (check page 86) command.

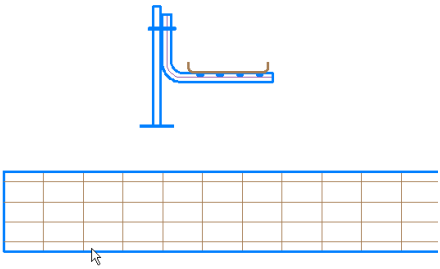
This command allows to draw single trays or automatic drawing of tray sequences with simultaneously insertion of elbows (or without elbows). Tees and reducers are also available.
These objects are automatically connected to trays (one–click technology).



For more convenient edition of trays with automatic insertion of elbows and tees the **Cablofil – Couple cable trays** command is used.

In order to enable the creation of mesh trays user can also find in the **Cablofil** library apart from the basic elements also categories that include:

Mountings scheme – this category contains schemes that represent the tray mounting. Mounting schemes are used only as graphical supplementary information. Because of that mounting symbols are not included when creating BOMs.
In order to automatically insert mounting schemes user can sue the **Create alternative view** command.



Accessories – this category includes additional elements, e.g. handles, anti fire compartments etc. Such elements inserted to the project will be included when creating BOMs. Some elements (e.g. clips, handles etc.) are too small to be visible on plans. Instead of inserting many "small" objects to the project it is possible to insert a table with additional elements. Elements with filled amount will be included in BOMs.

To edit the amount of additional elements user can use the command to edit the attributes or the **CADprofi Attributes and descriptions** command.

	BLF 8/16	CU	50
	SCMT	GS	100
	GRIFEQUIP	AL	100

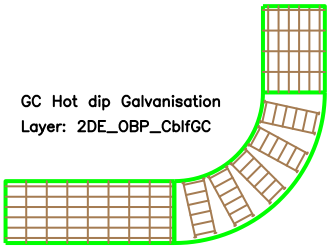
Couplers catalog – couplers data is included when drawing trays. Because of that there is no need to add to the drawing couplers as graphical objects. In the couplers catalog it is only possible to check the couplers technical data.

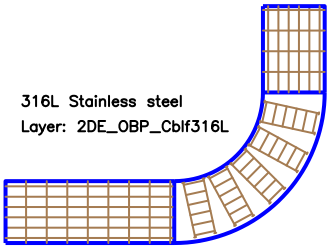
Trays material

During designing it is possible to specify the **Cablofil** mesh trays material. Program inserts trays that are created with different materials on separate layers.

For example the **2DE_OBP_CblfEZ** layer is used for **Cablofil** electrolytically galvanized (**EZ**), the **2DE_OBP_Cblf316L** layer is used for trays made of stainless steel (**316L**).

During insertion of **Cablofil** elements program checks the layer name and includes only elements that lie on layers that begin with the **2DE_OBP_Cblf**. Because of that user should not change the trays layer name.

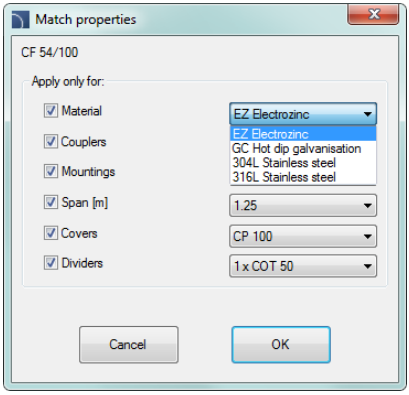




An exception of this rule may be for example a necessity to add additional red layer for anti fire or other special installations. For example the **2DE_OBP_CblfEZ_ppoz** can be used in anti fire installations for electrolytically galvanized trays. In such case in **Cablofil** BOMs instead of material, the layer name will be visible.

To change the material and other trays parameters user can sue the **Cablofil – Match properties** command.

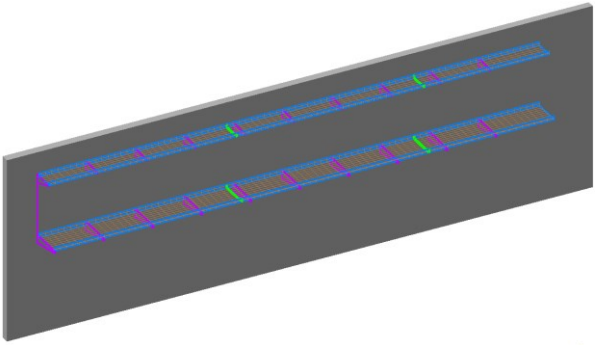
To avoid misallocation of tray material **it is not recommended** to change the trays layer with the CAD program standard options.



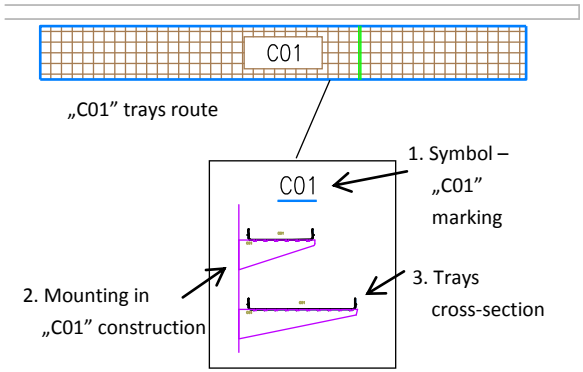
Complex constructions

In real installations it is very often needed to conduct trays one above the other. Sometimes such constructions can consist of many trays that are mounted to the ceiling, walls, etc. On plans it is very difficult to represent such constructions, because many trays that overlap each other, therefore the project becomes unreadable.

This problem is solved by "complex constructions". In this case instead of drawing many trays that overlap, it is possible to draw a route, which shows the trays sequence and a scheme that represents the arrangement of ducts and fittings in the route.



Example: two trays – one above the other



2D drawing – advanced construction in CabloCAD

Complex construction scheme must contain:

1. **Construction marking symbol** – marking must be unique (e.g. C01, A15 etc.). In one project it is not possible to define several complex constructions with the same marking (ID).
2. **Mounting** – scheme that contains special mounting symbols. When inserting mounting symbols the mounting spacing and construction ID must be provided (e.g. C01, A15…).
3. **Trays cross-section** – when inserting mounting symbols user should also specify the couplers type, thanks to which trays and construction markings are connected (e.g. C01, A15…).

Trays route – when drawing a tray route it is necessary to specify its total width and height that is occupied by all trays in the specified route.

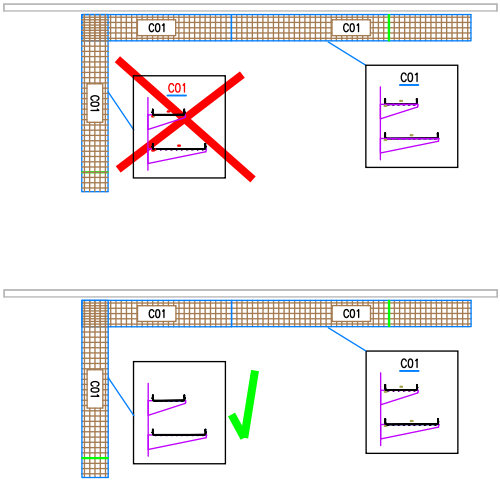
Notice:

To a specified complex construction (e.g. C01) belong all elements that contain the specified ID (e.g. C01). For this reason, for visual control, the ID is visible in all mounting symbols and trays cross-section. This ID is on a non-printed layer, that's why it is visible only on the screen.

In a definition of one complex construction (i.e. one design) it should be avoided to mix elements with different IDs. When copying elements between constructions it is necessary to arrange IDs (i.e. construction attribute marking).

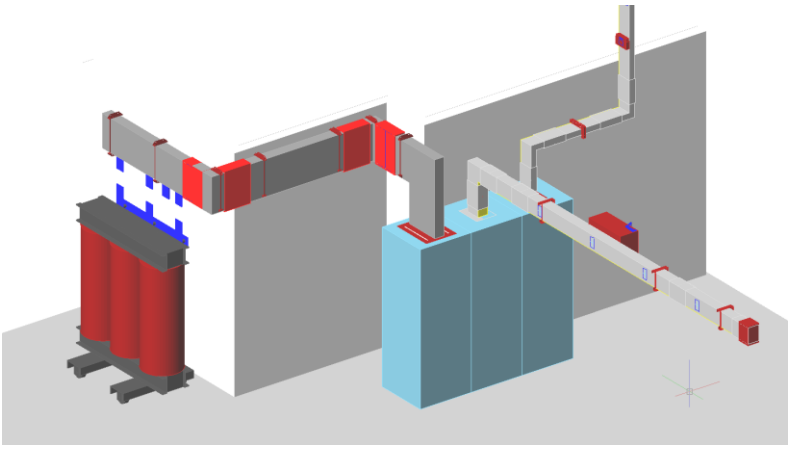
It is not possible to define several same complex constructions in a project. For this reason schemes with complex construction definition should not be duplicated. This can lead to errors when creating BOMs (duplicated elements in the drawing can duplicate the number of trays and accessories in BOMs).

If user needs to duplicate schemes of complex constructions, all copies should be "exploded" or they must have "cleared IDS" (empty marking attribute). These copies won't be included by the program in BOMs).



Designing busbars installations

Thanks to the **Legrand** or **PiN ENERGIA** library of products that are available in the **Manufacturers library** (check page 86) command it is possible to create projects of busbars installations both in 2D and 3D.

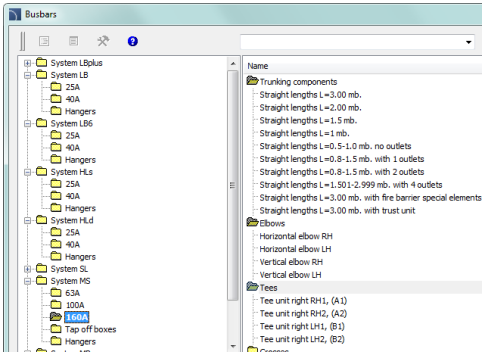


Example of a busbars installation project

> ***Hint***
>
> ***In order to receive a detailed bill of materials it is recommended to design busbars in 3D.***

Libraries content

In libraries we will find all elements that are necessary to design complete busbars installations. Transformers, switchgear and various accessories are also available.



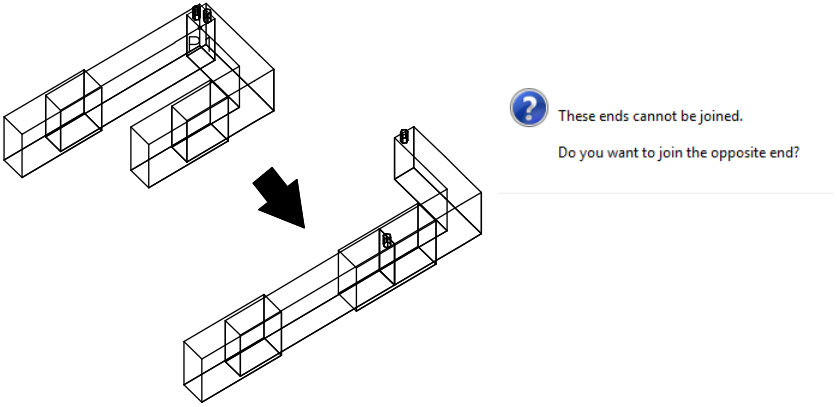
Inserting and connecting object on the drawing

Inserting objects is intuitive, and the library usage does not differ from general principles used in designing in **CADprofi** program.

All busbars have appropriately defined connection points, thanks to which application can automatically connect inserted object with an object that is already in the drawing. This process makes it faster to design installations, especially in 3D view, freeing the user from the hassle of determining the insertion points and rotation angles. Program also verifies the correctness of connections, thus ensuring the correct execution of the project.

Designing busbars installations is based on insertion of elements with automatic connection to the elements that were previously inserted. This process is repeated until the whole installation is prepared. This design model is very smooth and quick (even in 3D), and also includes the required precision.

When choosing objects user should pay attention to the **Insertion point** settings. Appropriate selection of insertion point allows to automatically connect objects. In the event of incorrect connection, a message is displayed informing about the lack of possibility to connect the busbars ends, giving the possibility to rotate the connected item.



Correct object connection

Notice

Automatic object connection is available only for objects that fit to each other – usually within the same system.
When connecting, the end of the busbar is chosen that was located closer to the point indicated by the user.

Busbars

CADprofi® Mechanical

CADprofi Mechanical

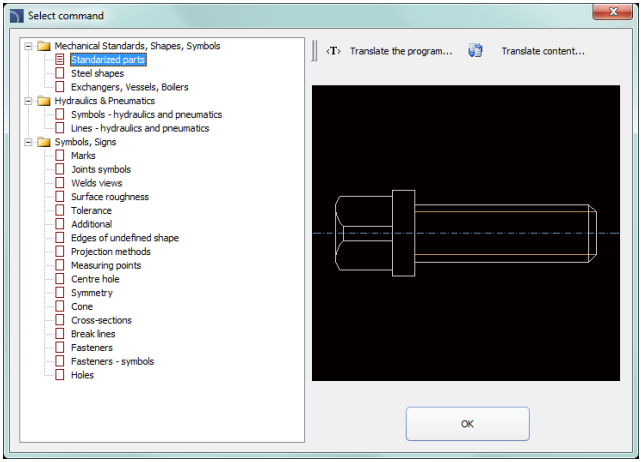
CADprofi Mechanical – Introduction

Mechanical module helps with designing mechanical and construction projects in 2D.
This module has got a very extensive library of standardized parts, steel shapes and other steel products developed on the basis of national and international standards. These include bolts, nuts, washers, pins, heat exchange elements and many other elements. Scheme editor allows to easily design hydraulic and pneumatic systems.

CADprofi Mechanical in 2 minutes

Mechanical – general commands

The **Mechanical** command opens the dialog window from which we can choose the appropriate commands from the **Mechanical** module. This command has been prepared for beginners. In the dialog window apart from the list of all commands we can also find a large preview window that shows the outline of the selected function.



Mechanical command dialog window

Procedures

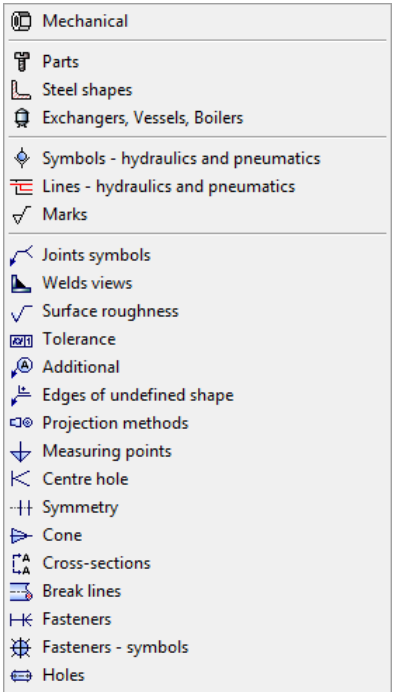
Selecting the command

1. Run the **Mechanical** command.
2. In the dialog window, expand the chosen category and select the needed command.
3. Selected command will start after clicking the **OK** button.

CADprofi Mechanical menu and toolbar

CADprofi Mechanical commands

CADprofi Mechanical toolbar



Menu of CADprofi Mechanical module

Editing commands

Important editing commands used in designing can be found in the **CADprofi** menu and **CADprofi – Tools** toolbar.

CADprofi Tools toolbar

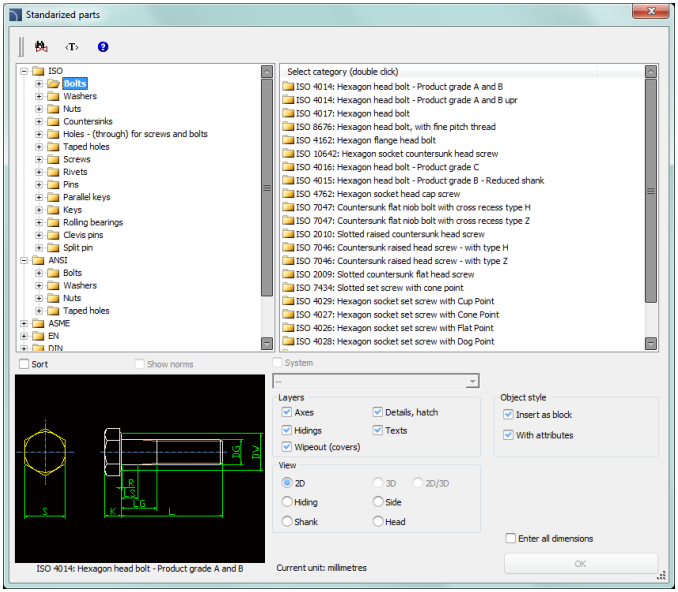
CADprofi Mechanical: main program commands

Standardized parts

The **Standardized parts** command allows to insert to the drawing standardized elements such as: screws, nuts, rivets, keys, bearings, wedges etc. All objects has got different views, which allows to create cross-sections, hidings etc.

For all objects there are databases available with many national and international standards, including: **PN**, **EN**, **ISO**, **DIN**, **GOST** and **ANSI**. It is possible to specify which standards should be available to the user. To do this, user should select which standards should be visible in the **CADprofi - Options** command (in the **Standards available in the program** card).

All elements are parametrical objects. It means that during inserting them into the drawing user can specify all dimensions. This feature gives user a lot of opportunities, because with the possibility to determine the dimensions, he can use any devices even those that are not in the standard program databases.



Standardized parts dialog window

The **Standardized parts** dialog window contains the following items:

Categories tree – a list of all available elements divided into categories (standards) and subcategories.

Detailed list – contains a list of available element types with various dimensions.

Preview – displays the preview of the currently selected element. Right-clicking on the preview window will zoom the view.

Sort – enables/disables categories alphabetical order.

Show norms – enables/disables displaying symbols according to standards.

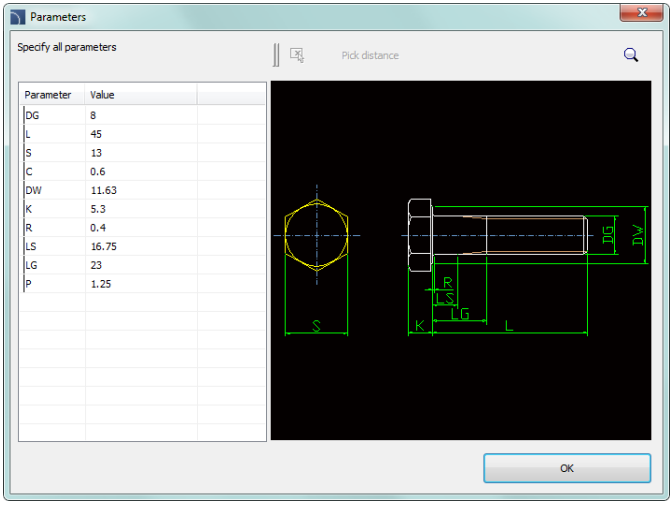
View– object view selection. Available options depend on the selected element.

Object style – options that allow to specify the object style and the level of object detail.

- **Insert as block** –allows user to insert an element that is exploded into component parts or a one single object (block).
- **With attributes** – enables/disables adding of attributes to the inserted object.
- **Axes** – enables/disables an option to draw axis in the inserted object.
- **Hidings** – enables/disables drawing elements that are hidden in an object.
- **Wipeout (covers)** – enables/disables addition to the inserted elements **Covers** (WIPEOUT) type objects.
- **Details, hatch** – enables/disables drawing of hatches and object details such as holes in the flanges, arrows that indicate the direction flow and others.
- **Texts**– enables/disables drawing of potential texts in objects.

Specify all parameters – insertion of selected object with the possibility to specify all editable parameters. Selecting this option and pressing the **OK** button will open a new **Parameters** window.

The **Parameters** dialog window allows to specify the values of each parameter. It is also possible to specify dimensions directly from the drawing. In such case user has to activate the dimension field and press the **Pick distance** button.



Parameters dialog window

> ***Hint***
>
> ***Selected object data is displayed as the default data in the „Parameters" dialog window.***

Procedures

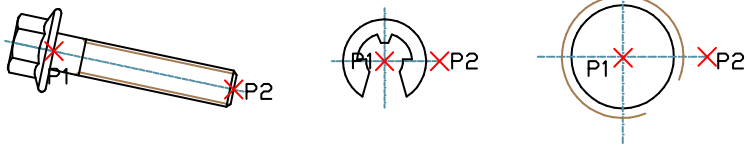
Inserting objects

1. Run the **Standardized parts** command.
2. In the dialog window select the needed object category. After that the element preview will be displayed and on the detailed list you will be able to see all available element types and sizes.
3. (Optional) If the list of available types possesses subcategories double click on a category will expand it and you will be able to see its content.
4. (Optional) Right click on any item and from the menu choose the **Expand all/Collapse all** option.
5. Choose the needed element form the detailed list.
6. (Optional) Change the **View** for the selected element.
7. (Optional) Change the settings for the **Object style** options.
8. (Optional) Select the **Enter all dimensions** option to specify inserted objects dimensions.
9. Click the **OK** button to insert selected object into the drawing.
10. Specify the insertion point.
11. Specify the object rotation angle.

Normalized parts

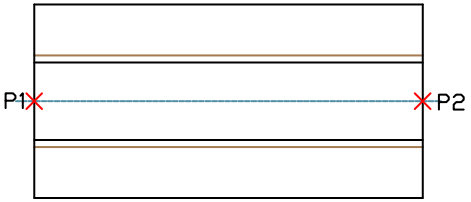
Inserting standardized parts to the drawing

The way how to insert selected symbols into the drawing depend on its type and the chosen view. Inserting objects, which dimensions are determined on the basis of types of series only need to select the insertion point (**P1**) and specify the rotation angle (**P2**).



During insertion of objects such as: holes (transitional, threaded), drills, screw joints, rivets and others user should specify an additional parameter - most often the element length (e.g. the dimension that determine the hole depth or the connection thickness).

Normally user should specify the (**P1**) point, and afterwards another point that at the same time define the object's length or other dimension (**P2**). This point will also determine the insertion angle.



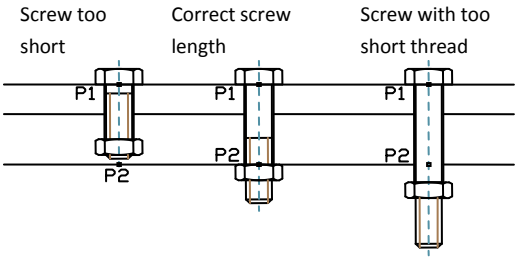
This parameter can be also specified by using the **Enter all dimensions** option. After specifying all dimensions in the dialog window, user should insert the object by indicating the insertion point (**P1**) and the rotation angle (**P2**).

Screw connections

The **Standardized parts** command allows to insert connections (simultaneously insertion of a screw and a nut) in three most commonly used variants:

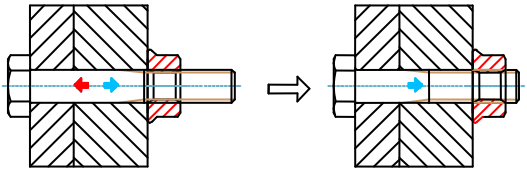
- hexagon head bolt + nut,
- countersink + hexagon socket head cap screw and nut,
- hexagon socket head cap screw + nut.

In order to insert a screw connection user should select the appropriate connection type and the screw size in the dialog window. Afterwards he should indicate on the drawing the insertion point (**P1**) and the connection thickness (**P2**).
For screw connections program controls the minimum and maximum connection length for the selected screw type and places a nut only in the thread length. Because of that, in connections it is necessary to select the appropriate screw length.
A mistake would be to insert for example a screw with a 30 mm length for a connection that has got a 40 mm thickness.



Quick edition

The **Quick edit** (check page 61) command allows dynamic lengthening or shortening of some standardized elements. For example: if in the drawing a M8 x 55 screw from the PN EN 24016 standard is inserted, then it can be lengthened or shortened to the length of 40 to 80 mm according to the elements contained in the specified standard.

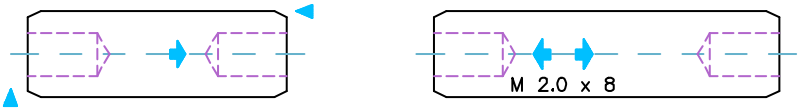


Example of a quick edition length change.

Notice

The disappearance of one of the tags means reaching the limit of the given types of series length.
Lack of tags mean that the object can't have the length edited.

After selecting the **Quick edit** command and indicating the object, arrows that symbolize the object enlargement and reduction will be visible. Each time the user clicks on one of them will replace the object to a bigger/smaller and update the information about its size.



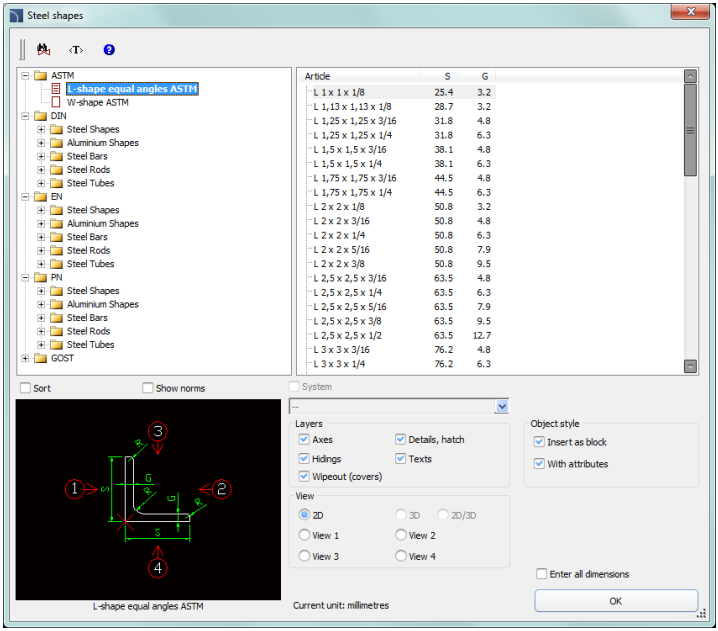
Procedures

Quick edition

1. Run the **Quick edit** command.
2. Indicate an object that was inserted with the **Standardized parts** command.
3. Edit an object by clicking on the lengthen/shorten arrows symbol.
4. End the command with the right mouse button or the **Enter** key.

Steel shapes

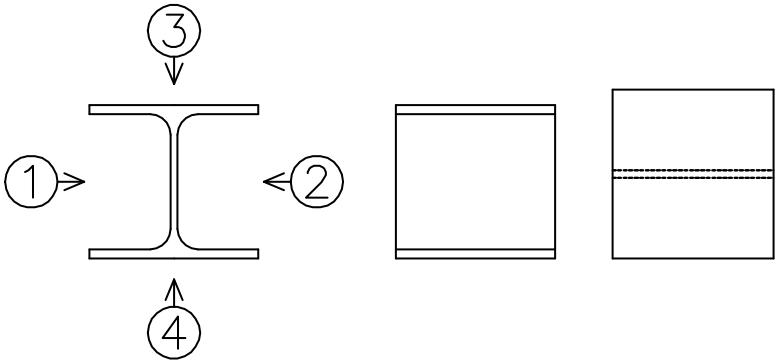
The **Steel shapes** command allows user to insert to the project all kind of steel shapes such as: flat bars, L-shapes, U-shapes, Z-shapes etc. For all objects databases with many national and international standards including: **PN**, **EN**, **DIN** and **ANSI** are available.
Steel shapes have got their own mass unit defined. Thanks to this it is possible to obtain information about length and mass of elements inserted into the drawing. It is also possible to automatically obtain the full BOM of steel (or other materials).
Most steel shapes available in **CADprofi** program are steel elements. Thanks to the possibility to specify all dimensions it is possible to insert custom profiles such as aluminum, copper and others. If custom profiles are being inserted then it is necessary to adjust the type, material kind and unit mass or the total mass of inserted elements. The easiest way to adjust these parameters is to use the **Attributes and descriptions** command which allow to simultaneously change the data for multiple objects (check page 34).



Steel shapes dialog window

Options used in this dialog window have been described in previous chapters (check page 295).
Steel shapes available in the library have got the possibility to be inserted in a main view or in additional views. These views are indicated in the preview window with arrows.

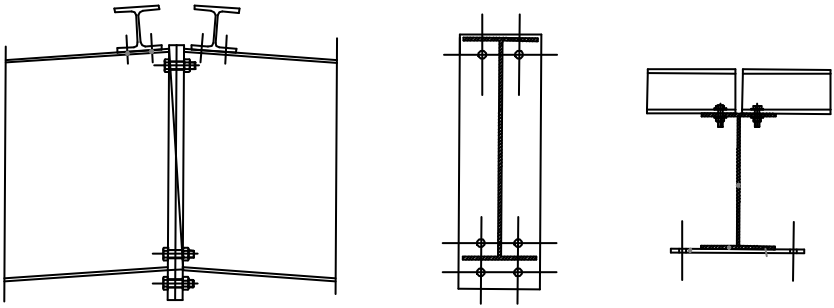
For some views, hiding lines are included. With the help of the **Object style** option it is possible to enable or disable the creation of invisible lines in the inserted object.



Procedures

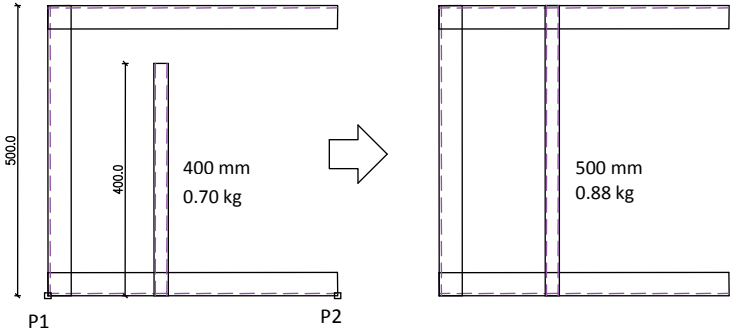
Inserting steel shapes

1. Run the **Steel shapes** command.
2. In the dialog window select the needed object category. After that the element preview will be displayed and on the detailed list you will be able to see all available element types and sizes.
3. (Optional) If the list of available types possesses subcategories double click on a category will expand it and you will be able to see its content.
4. (Optional) Right click on any item and from the menu choose the **Expand all/Collapse all** option.
5. Choose the needed element form the detailed list.
6. (Optional) Change the **View** for the selected element.
7. (Optional) Change the settings for the **Object style** options.
8. (Optional) Select the **Enter all dimensions** option to specify inserted objects dimensions.
9. Click the **OK** button to insert selected object into the drawing.
10. (Optional) If you decide to insert a view from the top, then in the **Parameters** window specify the select steel shape length (it is necessary to later create correct BOMs).
11. Specify the insertion point on the drawing.
12. If you decide to insert the main view specify the rotation angle. For other views, indicate the second point thanks to which you will determine the steel shape length.



Examples of drawing with inserted steel shapes

Steel profiles are inserted as blocks. X scale for side views correspond to the profiles length. In case of cross-sections (main views), the profile length corresponds to the block Z scale. Thanks to this by changing the scale (X or Z) in the properties window it is possible to adjust the profile length after inserting them into the drawing.



Closed profile with 400 mm length.

Changing the profile length to 500 mm, e.g. changing the X scale = 500

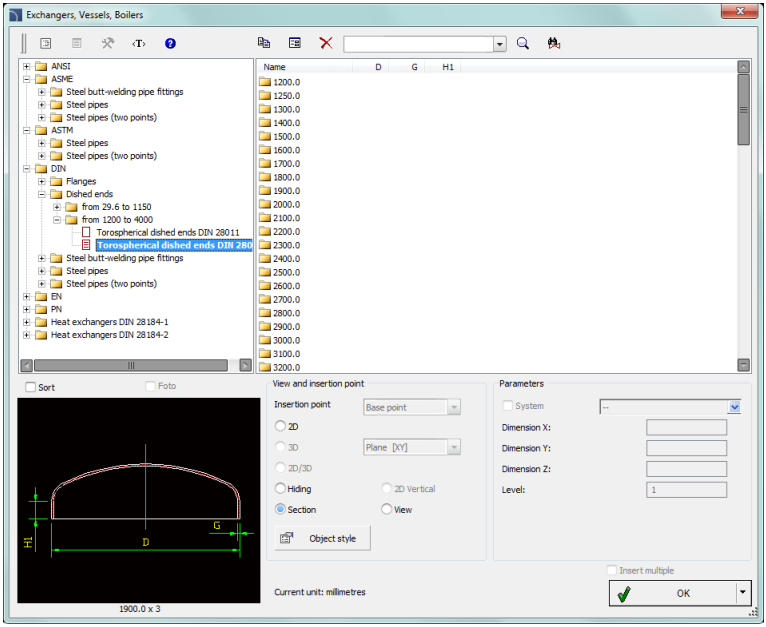
> ***Hint***
>
> ***To display the properties window in CADprofi program, user should press the CTRL + 1 or select the appropriate command from the CAD program top menu.***

Steel shapes and metallurgical products

Exchangers, pipes, fittings

The **Exchangers, pipes, fittings** command contains many elements that are used in designing heat exchangers, vessels, boilers etc. Elliptical bottoms, flanges, pipes, elbows, reducers and other elements are available. Most elements can be inserted into the drawing in side, top and cross-section views.

A unique program feature is the possibility to calculate the optimal distribution of the heat exchange according to the DIN 28184–1 and DIN 28184–2 standards. It is also possible to save the user dimensions by Creating and extending the user type of series **Creating and extending user type of series** (check page 88).



Exchangers, pipes, fittings dialog window

Options covered in this dialog window were described in previous chapters (check page 295).

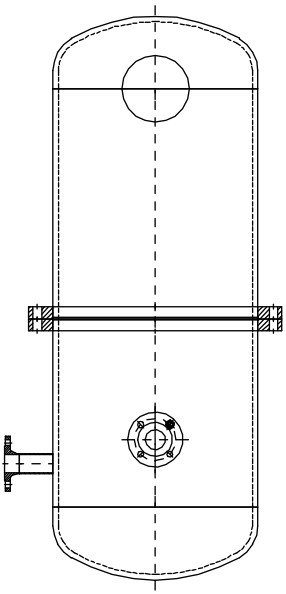
Procedures

Inserting apparatus parts

1. Run the **Exchangers, pipes, fittings** command.
2. In the dialog window select the needed object category. After that the element preview will be displayed and on the detailed list you will be able to see all available element types and sizes.

3. (Optional) If the list of available types possesses subcategories double click on a category will expand it and you will be able to see its content.
4. (Optional) Right click on any item and from the menu choose the **Expand all/Collapse all** option.
5. Choose the needed element form the detailed list.
6. (Optional) Change the **View** for the selected element.
7. (Optional) Change the settings for the **Object style** options.
8. (Optional) Select the **Enter all dimensions** option to specify inserted objects dimensions, for example a pipe.
9. Click the **OK** button to insert selected object into the drawing.
10. Specify the insertion point on the drawing.
11. Specify the object rotation angle. For certain objects (e.g. for pipe views) specify the second point thanks to which you will determine the length.

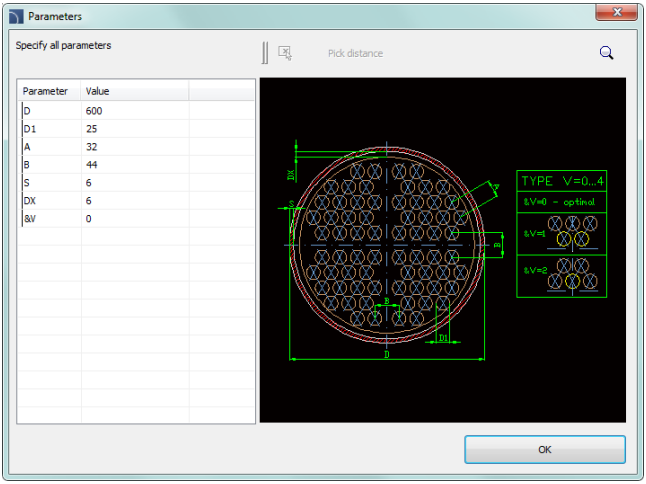
Designing apparatus



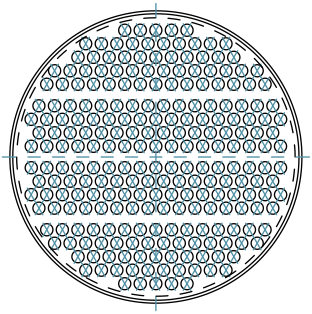
Example of an apparatus that consists of: bottoms, shells, nipples and flanges

When insert heat exchangers it is possible to select different flow direction according to the DIN standard. Apart from that it is possible to specify any dimensions for spacing and other arrangement of exchanger.

After specifying the diameter of the heat exchanger shell, program will select the optimum distribution of tubes that form the part of the exchange area.
It is also possible to make calculations after specifying the total amount of tubes. In this case program will calculate the minimum diameter of the heat exchanger shell and draw the optimal distribution of the exchange area.
Thanks to calculation user will receive the information about the total exchange distribution area and the number of tubes in each heat exchanger shell.



Specifying the heat exchanger parameters



Eingabeaufforderung	Wert
Name	Heat exchanger DIN 28184-1 - 4 beams (crosswise)
Article	Exchanger DN600
Manufacturer	DIN 28184-1
D2	571.5
Area	19.164
Element count	Amount of pipes: 244 (4x61)

Calculations result and the heat exchanger corss-section

Symbols and markings

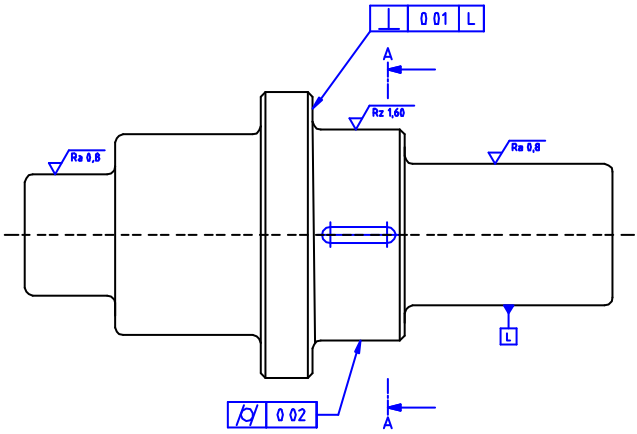
CADprofi program has got typical markings that are used to create mechanical branch documentation. These markings are prepared in accordance with national and international standards, including **PN**, **EN**, **ISO** and **DIN**.

Markings are multivariant elements, which appearance and information can be in a quick and easy way customized. In this way, one multivariant symbol can replace many "traditional" symbols in the library, giving the user bigger possibilities to configure the type and appearance. Multivariant symbols can be freely modified with the **Edit symbols** command.

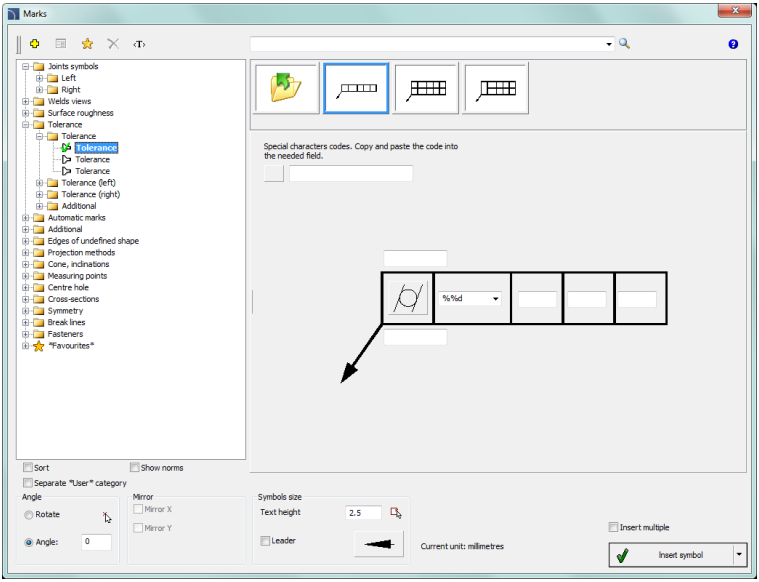
All marking are available in the **Marks** command. These symbols are also available in several additional commands, thanks to which user has got a quicker access to markings or symbols that he needs at the moment.

Commands used to mark technical drawings:

- Welds views
- Surface roughness
- Tolerance
- Edges of undefined shape
- Projection methods
- Measuring points
- Cross-sections
- Break lines
- Fasteners
- Holes
- and others.



An example of a drawing with various markings



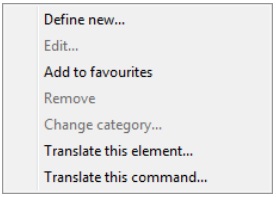
Marks dialog window

Marks dialog window contains the following items:

Tools menu:

- **Define new** – allows to add a new element to the database. Read more about it in the Defining user blocks (check page 89).
- **Edit** – allows to Editing user block parameters (check page 92).
- **Add to favourites** – copies symbol to the *Favourites* (check page170).
- **Delete** – deletes the selected symbols from the ***Favourites*** category.

Pop-up menu– options menu for the selected symbol, which is available after right clicking on the mouse button on the selected symbol. Commands in this menu are similar to the tools menu.



Categories tree – a set of all categories and symbols in the library, shown as a pull-down tree. User can customize the display order by using the following options:

- **Sort** – alphabetical order of categories placement and their content.
- **Show norms**– enables/disables displaying symbols according to norms.

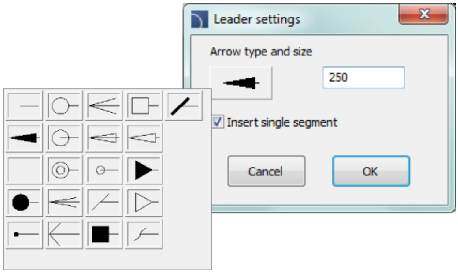
Thumbnail view – displays the content of currently selected category as thumbnails.

Angle/Rotation – possibility to specify a fixed angle when inserting an element or enabling the rotation option, in which element's angle is defined by rotation during insertion.

Scale – specifies the size of inserted symbols. It is possible to enter the scale value or to indicate a symbol on the drawing whose scale user would like to use.

Mirror X, Y– enabling this option will create an X or Y mirror image for certain specified object.

Leader – enables/disables usage of leader for descriptions. By clicking on the button, that symbolizes the currently set arrow type, user can change the following leader settings:

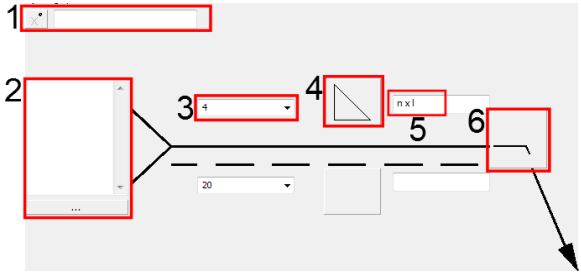


- **Arrow type**– symbol that is inserted in the leader starting point for example an arrow.
- **Arrow size** – determines the leader scale.
- **Insert single segment** –drawing of a one or multi-segment leader.

Insert multiple– enables/disables the possibility of inserting multiple symbols to the drawing. The insertion process requires from the user to press the **Enter** or **Esc** key when he finished inserting multiple symbols.

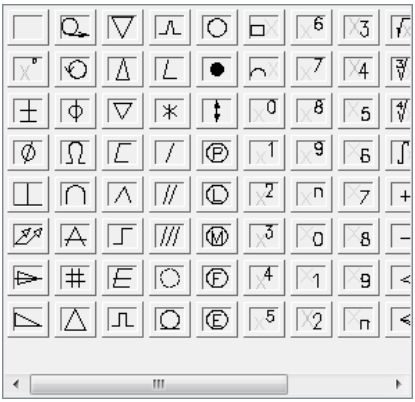
Text height – value that determines the text height and the whole block scale.

Symbol preview – an active area than contains the preview and parameters of selected marking. Each controls contained in this area give the possibility to customize the content of the multivariant symbol.



Examples of light control for multivariant symbols

1. **Special symbols codes field** – gives the possibility to display the code of the selected symbol so it can be copied and pasted anywhere into a text field. List of symbols is available after pressing the button.



2. **Text field** – gives the possibility to enter any multiline text. This field can be filled with any information selected from the additional window that is opened with the **Browse** button.

3. **Pull-down list** – gives the possibility to enter any value or to select a value from the list (by pressing the button). There are also lists available that allow only to select the value (with the possibility to edit the field).

4. **Graphical pull-down list** – gives the possibility to select one of available graphical markings.

5. **Editable field** – gives the possibility to enter any text.

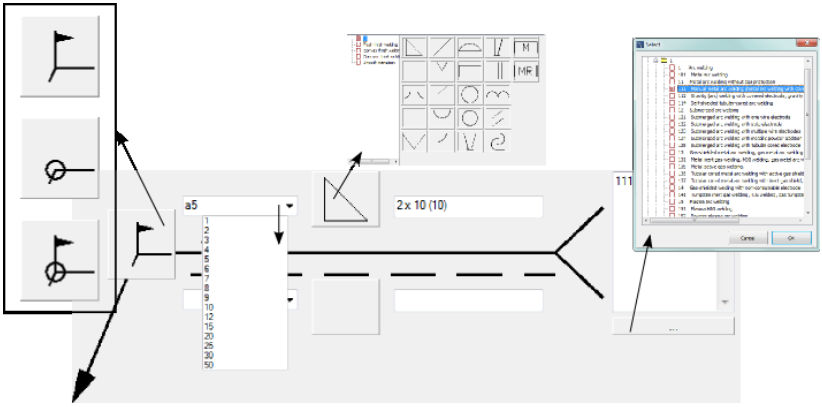
6. **Supplementary symbol button** – every time the user clicks on this button will display the next available symbol in the button field.

For some multivariant symbols, other controls may also be available, such as buttons used for measuring dimensions or angles in the drawing, enable/disable fields and more.

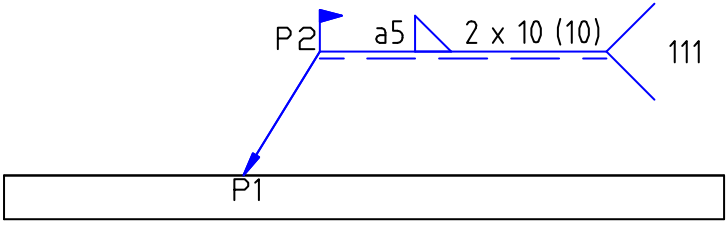
Procedures

Inserting a fillet weld marking

1. Run the **Joint symbols** command.
2. In the dialog window select the needed symbol.
3. Click the supplementary symbol button in order to get the **Mounting weld** symbol.
4. Specify the weld thickness (e.g. **a5**) or select the value from the pull-down list.
5. Click the icon of the weld symbol and from the newly opened dialog window select the fillet weld symbol.
6. Enter the weld length e.g. **2 x 10 (10)**.
7. Click the browse button and the dialog window select the process of weld preparation e.g. **111 Manual metal arc welding**.
8. Enable the **Leader** option.
9. Click the **OK** button to insert the selected symbol into the drawing.
10. Specify the point that will be the leader beginning (**P1**).
11. Specify the symbol insertion point (**P2**) and possible rotation angle.



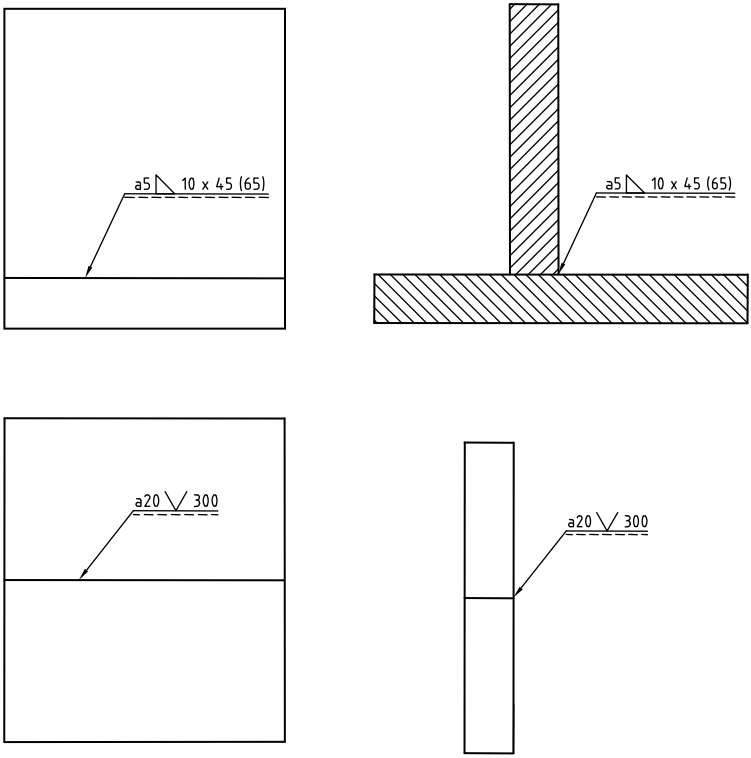
Settings in the dialog window



Joint symbols

The **Joint symbols** command contains markings according to the ISO 2553 standard. Descriptions of process connections are in accordance with the ISO 4063 standard. Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

Below are examples of using this command:



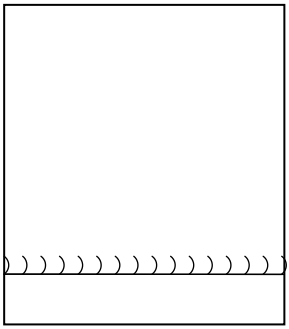
Joint symbols

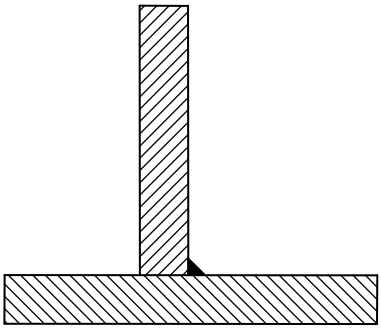
Welds views

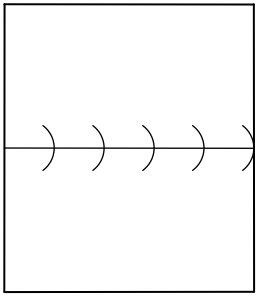
The **Weld views** command contains markings according to the ISO 2553 standard.

Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

Below are examples of using this command:







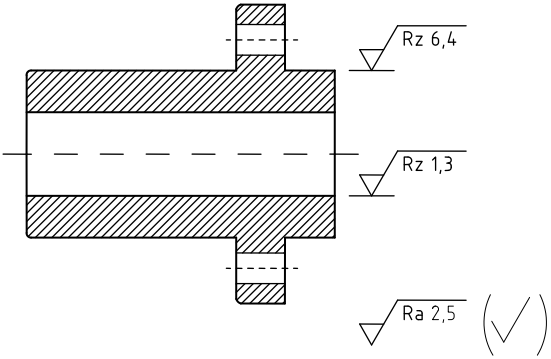
Welds views

Surface roughness

The **Surface roughness** command contains markings according to the ISO 1302 standard.

Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

Below are examples of using this command:

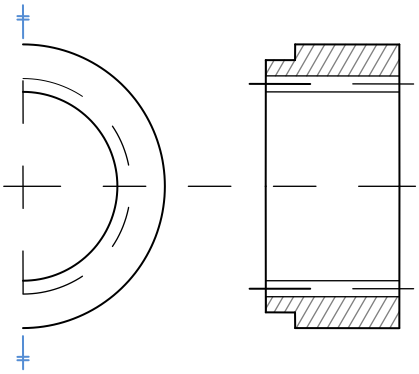


Symmetry

The **Symmetry** command contains markings according to the ISO 128 standard.

Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

Below are examples of using this command:

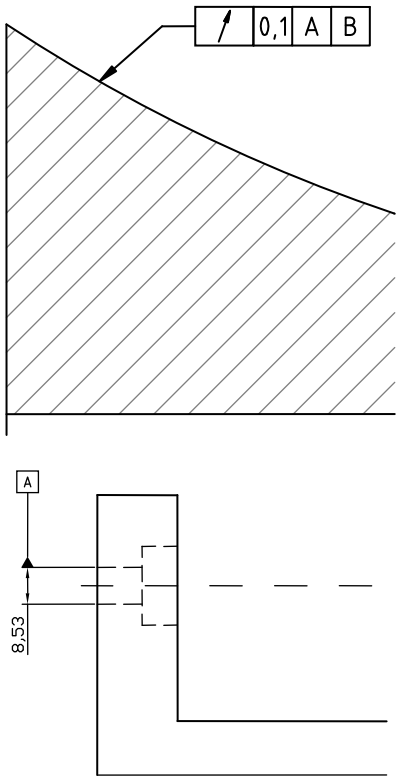


Tolerance

The **Tolerance** command contains markings according to the ISO 1101 standard.

Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

Below are examples of using this command:

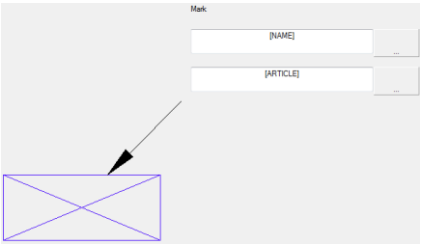


Additional markings

The **Additional** command contains various markings that are often used in technical drawings e.g. different variants of reference lines, position numbering symbols and universal markings that gather values from drawing objects.
Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

Numbering symbols allow to number objects and to gather parameters from them (e.g. name, type etc.). In this way it is possible to quickly number many objects and place the require texts in the drawing.

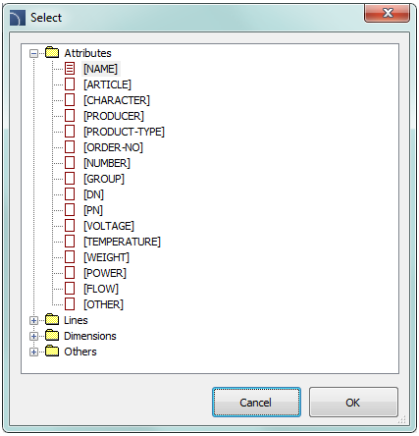
CADprofi program automatically detects numbered objects in the drawing, thanks to which in the dialog window the first available number from the specific branch (e.g. ME – mechanical) is being displayed. The button allows to select the next highest number



By adding different number prefixes, user can create many numbered sequences (e.g. one sequence without prefixes **1**, **2**, **3**…., and second **P1**, **P2**, **P3**).
Inserting symbols with the **Insert mupltiple** option will cause the symbols to have numbers added to them with increased numeration.
The **Mark 1**, **Mark 2** fields allow to add in the selected symbol any text or information retrieved from the indicated object. Selecting the information type will be possible after pressing the button that is located next to the field.

In the newly opened window user can select the following information:

- **Attributes** – indicated object attributes values.
- **Lines** – line type or installation kind.
- **Dimensions** – dimensions retrieved from the indicated object.
- **Others** – for example the CAD program system variable values.

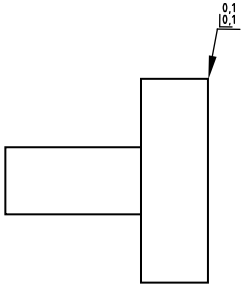


Edges markings

The **Edges of undefined shape** command contains markings according to the ISO 13715 standard.

Functionality and the detailed way of handling the command are described in the Symbols and markings (check page 306) chapter.

Below are examples of using this command:

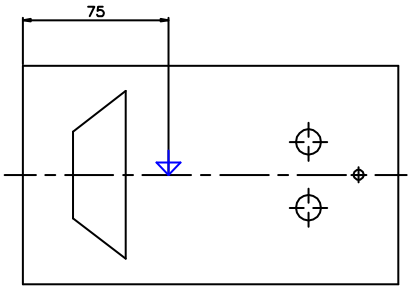


Measuring points

The **Measuring points** command contains markings according to the ISO 128 and DIN 6773 standards.

Functionality and the detailed way of handling the command are described in the Symbols and markings (check page 306) chapter.

Below are examples of using this command:

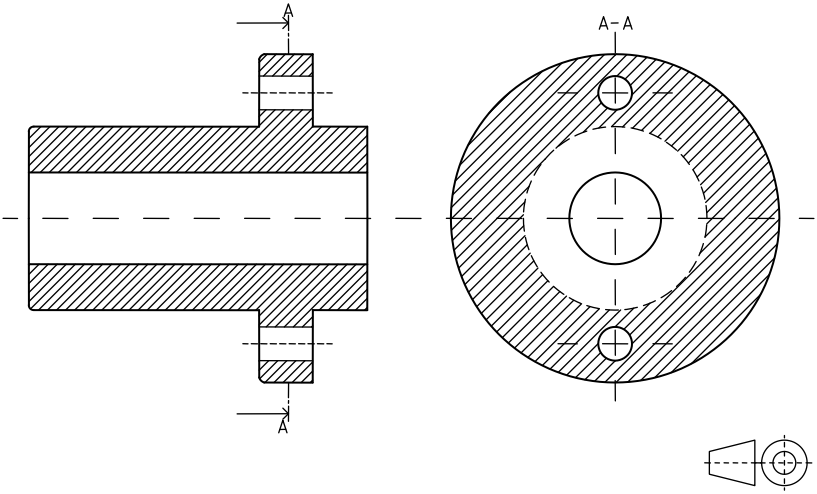


Projection methods

The **Projection methods** command contains markings according to the ISO 5456 standard.

Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

Below are examples of using this command:

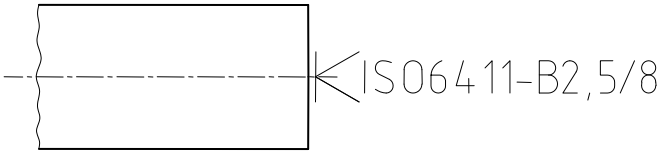


Centre hole

The **Centre hole** command contains markings according to the ISO standards.

Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

Below are examples of using this command:

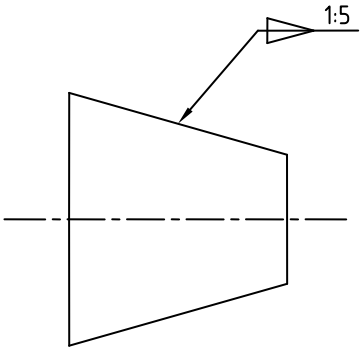


Cone

The **Cone** command contains markings according to the ISO 3040.

Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

Below are examples of using this command:

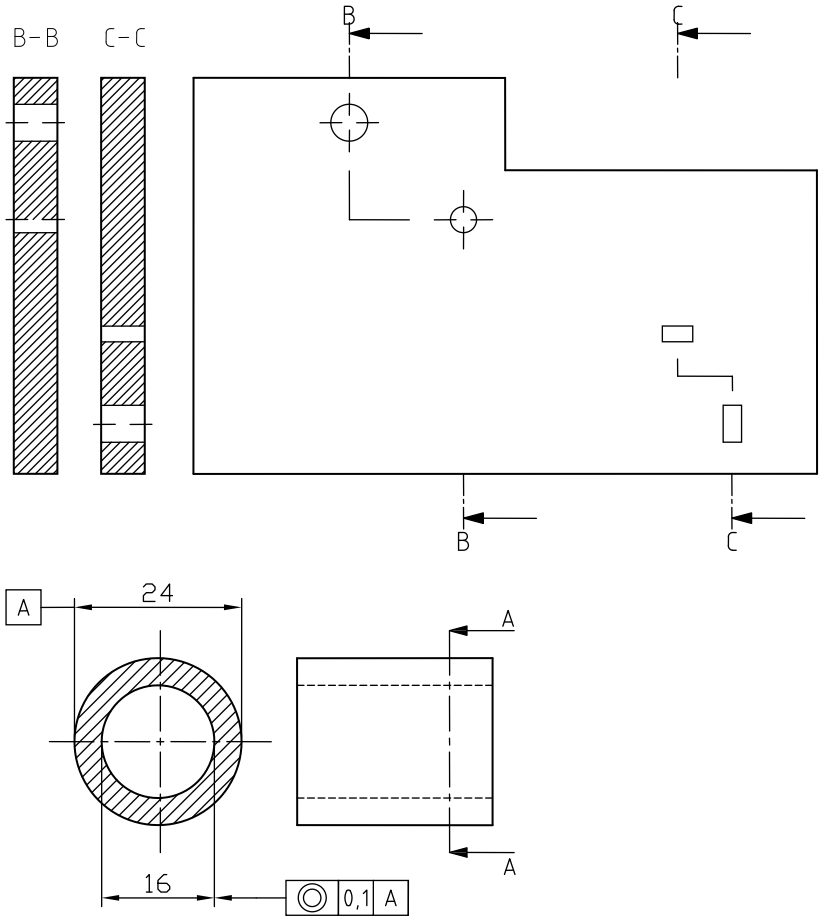


Cross-sections

The **Cross-sections** command contains markings according to the ISO 128 standard.

Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

Below are examples of using this command:



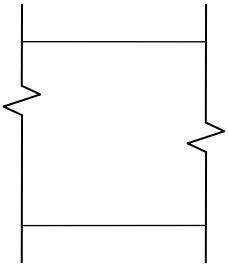
Line break

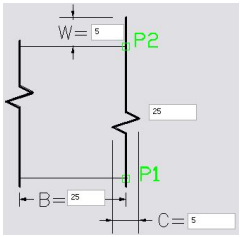
The **Line break** command contains various object line breaks.

Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

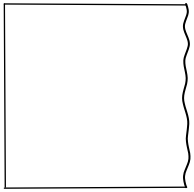
Below are examples of using this command:

Zigzag line

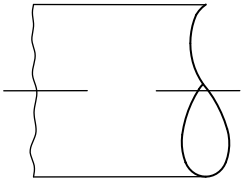


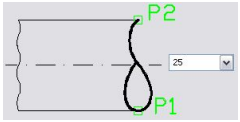


Wavy line



Pipe, cylinder interruption





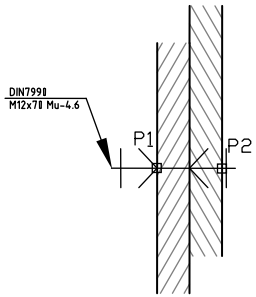
Fasteners

The **Fasteners** command contains markings according to the ISO 5845, as well as PN–81/N–01613 and PN–83/N–01635 standards.

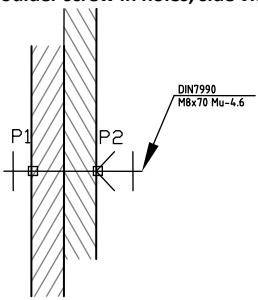
Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

Below are examples of using this command:

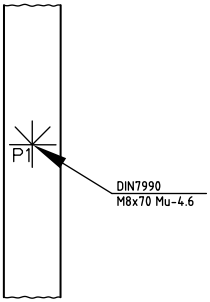
Screws

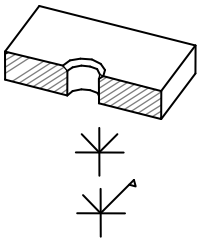


Shoulder screw in holes, side view

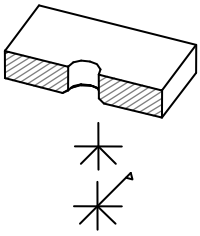


Shoulder screw in holes, top view



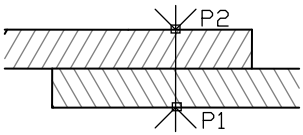


Proximal side deepening

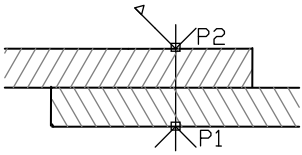


Other side deepening

Holes – workshop drilling



Holes – terrain drilling



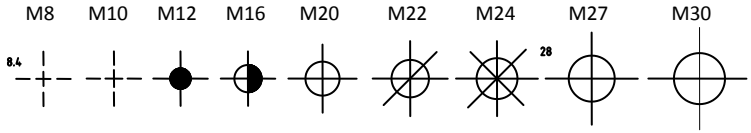
Fasteners symbols

The **Fasteners - symbols** command contains various symbols of rivets and screws.

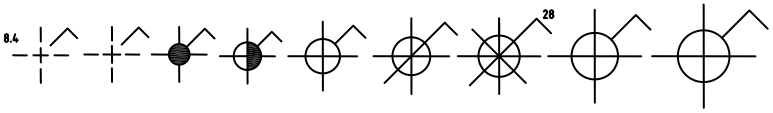
Functionality and the detailed way of handling the command are described in the Symbols and markings (check page 306) chapter.

Below are examples of using this command:

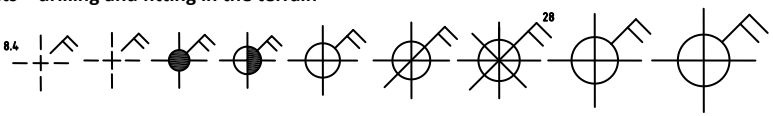
Rivets with round head on both sides



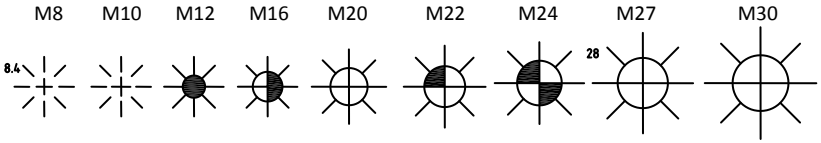
Rivets – riveting in the terrain



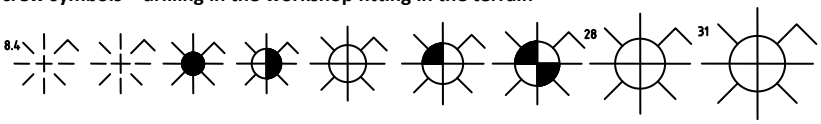
Rivets – drilling and fitting in the terrain



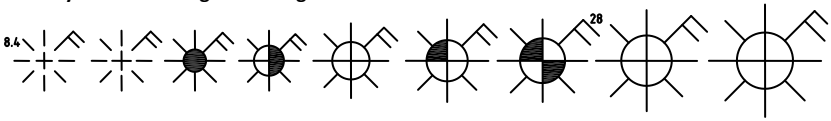
Screw symbols – drilling and fitting in the workshop



Screw symbols – drilling in the workshop fitting in the terrain



Screw symbols – drilling and fitting in the terrain

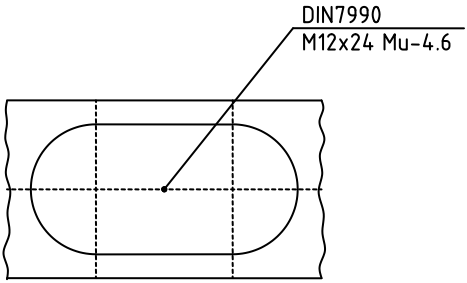


Holes

The **Holes** command contains markings according to the ISO 5261 standard.

Functionality and the detailed way of handling the command are described in the **Symbols and markings** (check page 306) chapter.

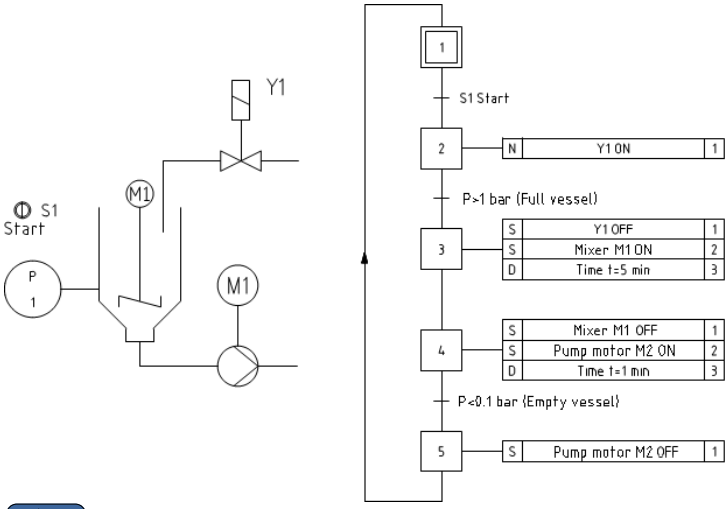
Below are examples of using this command:



Technological processes schemes

CADprofi program allows toc reate functional schemes, diagrams and technological processes schemes according to the EN 60848 standard.

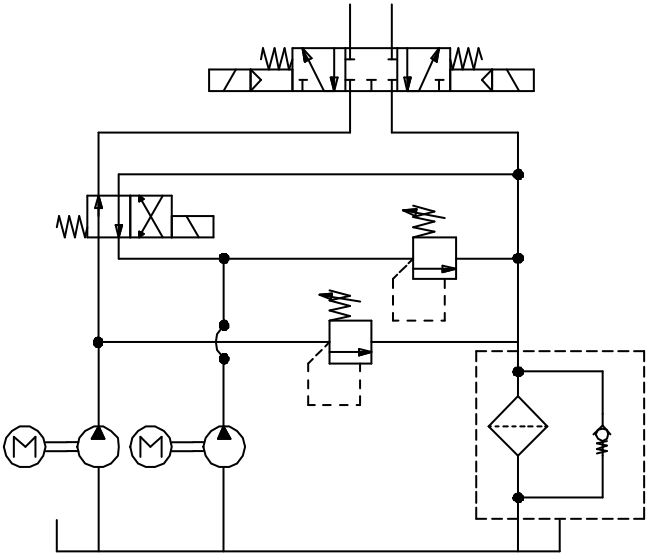
Check the **Flowcharts and diagrams** command description (check page 82).



Creating diagrams

Hydraulics and pneumatics symbols

The **Symbols - hydraulics and pneumatics** command contains elements that are used in creation of hydraulics and pneumatics schemes.
Creating installation schemes is associated with drawing lines, inserting symbols and describing drawings. **CADprofi** program gives to the designer, the appropriate commands, that make it easier to describe scheme elements, thanks to which it is possible to create BOMs from the symbols and lines that are used in projects.

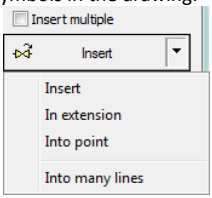


An exemplary scheme

All symbols available in **CADprofi** program are being inserted as blocks, which can be copied, deleted or edited with the use of standard CAD program commands. Most symbols can be also edited with the use of **CADprofi** editing commands such as: **Quick edit**, **Edit symbols** and **Delete symbols**.
Symbols are often used to graphically represent real devices, armature and other installations elements, therefore it is possible to add technical parameters, about the products used in the project, to the symbols (**Attributes and descriptions** command), as well as numbering and symbols' marks. This information can be used when describing drawings when creating graphical **legends or specifications** (check page 46).

Program possesses several options that make it easier to insert symbols in the drawing.

Main options (available after pressing the ▼ button):



- Insert symbol.
- Into many lines.
- In extension.
- Into point.

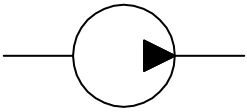
'Insert symbol' option

This is the default option that allows to insert symbols in any point or into a line indicated by user. When inserting symbols into lines, the symbol insertion angle depends on the line angle (the angle specified in the dialog box is being ignored).

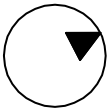
Procedures

Insert symbol

1. Select the symbols from any library.
2. Click the **Insert symbol** button. The library dialog window will close allowing insertion of symbol into the drawing.
3. Point into a line, in which you would like to insert the symbol.
4. (Optional) Click anywhere in the drawing to insert a symbol.
5. (Optional) Specify the rotation angle of inserted symbol - this option is available only if user has selected the **Rotate** option in the library dialog window.



Insert into a line



Insert in any point

'Into point' option

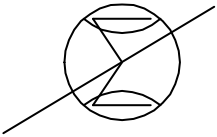
The **Into point option** may be used when it is necessary to insert the symbol in a point that lies on a line, but in such a way that the symbol won't "adjust itself to the line". In this option, the inserted symbol "ignores" the line, so it does not break it and it does not take angle from it. This option is used in many situations for example when inserting sensors, which usually need to touch lines.

Procedures

Insert into indicated point

1. Select the symbols from any library.
2. With the use of ▼ button, pull-down the insertion list.
3. Click the **Into point** button and insert the symbol into the drawing.

4. Point into a point in a line, in which you would like to insert the symbol.
5. (Optional) Click anywhere in the drawing to insert a symbol.
6. (Optional) Specify the rotation angle of inserted symbol - this option is available only if user has selected the **Rotate** option in the library dialog window.



Inserting a symbol without adjusting it into the line

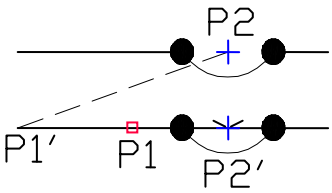
'In extension' option

The **In extension** option allows to precisely specify the symbol insertion point in relation to other objects located in the drawing. This allows to insert a symbol in a point in accordance with for example another symbol.

Procedures

Insert in extension

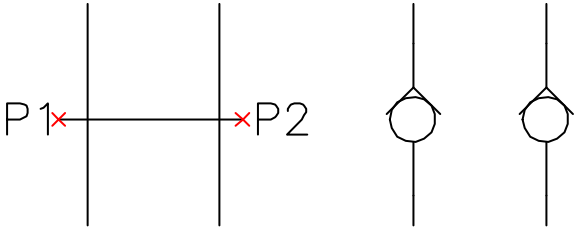
1. Select the symbols from any library.
2. With the use of button, pull-down the insertion list.
3. Click the **In extension** button. The library dialog window will close allowing insertion of symbol into the drawing.
4. Click a line, in which you would like to insert the symbol (**P1**).
5. Specify the symbol insertion point against you would like to specify the location of the inserted element (**P2**).
6. Specify an additional offset or click **Enter** to confirm the insertion point.
7. (Optional) Specify the rotation angle of inserted symbol - this option is available only if user has selected the **Rotate** option in the library dialog window.
8. Symbol will be inserted in the drawing and the command will end (if user has not selected the **Insert multiple** option).



Inserting in extension

'Into many lines' option

Into many lines option gives the possibility to quickly insert selected symbol into many lines. It allows to insert selected symbol into all lines that cross with the „indication line", that was specified during insertion (**P1–P2**).



Inserting symbols into many lines

Procedures

Insert into many lines

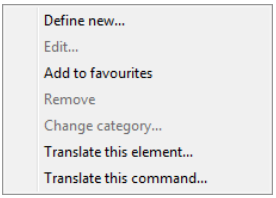
1. Select the symbols from any library.
2. With the use of button, pull-down the insertion list.
3. Click the **Into many lines** button. The library dialog window will close allowing insertion of symbols into the drawing.
4. Specify the first point of the "crossing line" that will cross with lines (**P1**).
5. Specify the second point of the "crossing line" that will cross with lines (**P2**).
6. The selected symbol will be inserted in all lines that crossed with the "crossing line".

The **Symbols - hydraulics and pneumatics** dialog window contains the following items:

Tools menu:

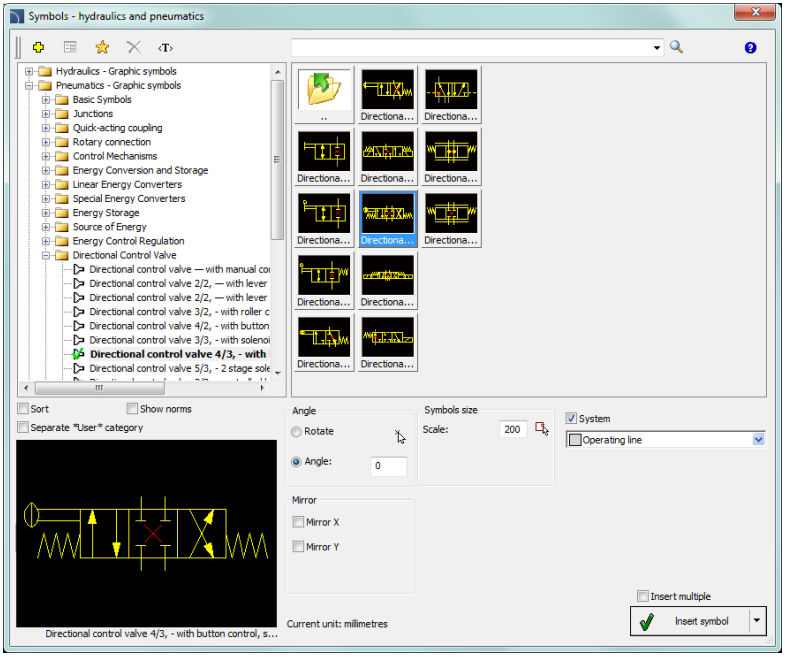
- **Define new** – allows to add a new element to the database. Read more about it in the **Defining user blocks** (check page 89).
- **Edit** – allows to **Editing user block parameters** (check page 92).
- **Add to favourites** – copies symbol to the ***Favourites*** (check page170).
- **Delete** – deletes the selected symbols from the ***Favourites*** category.

Pop-up menu – options menu for the selected symbol, which is available after right clicking on the mouse button on the selected symbol. Commands in this menu are similar to the tools menu.



Categories tree – a set of all categories and symbols in the library, shown as a pull-down tree. User can customize the display order by using the following options:

- **Sort** – alphabetical order of categories placement and their content.
- **Show norms**– enables/disables displaying symbols according to norms.



Symbols – hydraulics and pneumatics dialog window

Thumbnail view – displays the content of currently selected category as thumbnails.

Preview – displays the preview of the currently selected symbol. Right-clicking on the preview window will zoom the view.

Angle/Rotation – possibility to specify a fixed angle when inserting an element or enabling the rotation option, in which element's angle is defined by rotation during insertion.

Scale – specifies the size of inserted symbols. It is possible to enter the scale value or to indicate a symbol on the drawing whose scale user would like to use.

System – if in the program options, the extended layer name structure (check page. 29) is enabled then symbols will be inserted on layers depending on the kind of installation (system). In case of inserting symbols into an existing line then the **System** option is being ignored and the symbol layers depends on the line kind.

Mirror X, Y– enabling this option will create an X or Y mirror image for the specified object.

Insert multiple– enables/disables the possibility of inserting multiple symbols to the drawing. The insertion process requires from the user to press the **Enter** or **Esc** key when he finished inserting multiple symbols.

Procedures

Inserting symbol

1. Run the **Symbols - hydraulics and pneumatics** command from the toolbar or from the **CADprofi – > CADprofi Mechanical** menu.
2. In the dialog window choose the appropriate symbol.
3. Specify the insertion parameters: **System**, **Rotation**, **Scale**.
4. Click the **Insert symbol** button in order to specify the symbol insertion point.
5. (Optional) With the use of the button pull-down the insertion list and choose **In extension** to specify the insertion point in accordance to other objects.
6. (Optional) With the use of the button pull-down the insertion list and choose **Into point** to insert the symbol in a specific point.
7. (Optional) With the use of the button pull-down the insertion list and choose **Into many lines** to insert symbols simultaneously into many lines.
8. Click on a point in the line to insert the symbol.
9. (Optional) Click on any point in the drawing, to insert the symbol.
10. (Optional) Specify the inserted symbol angle - this option is available only if the **Rotation** option in the library dialog window has been enabled.
11. Symbol will be inserted in the drawing and the command will end (if user has not selected the **Insert multiple** option).

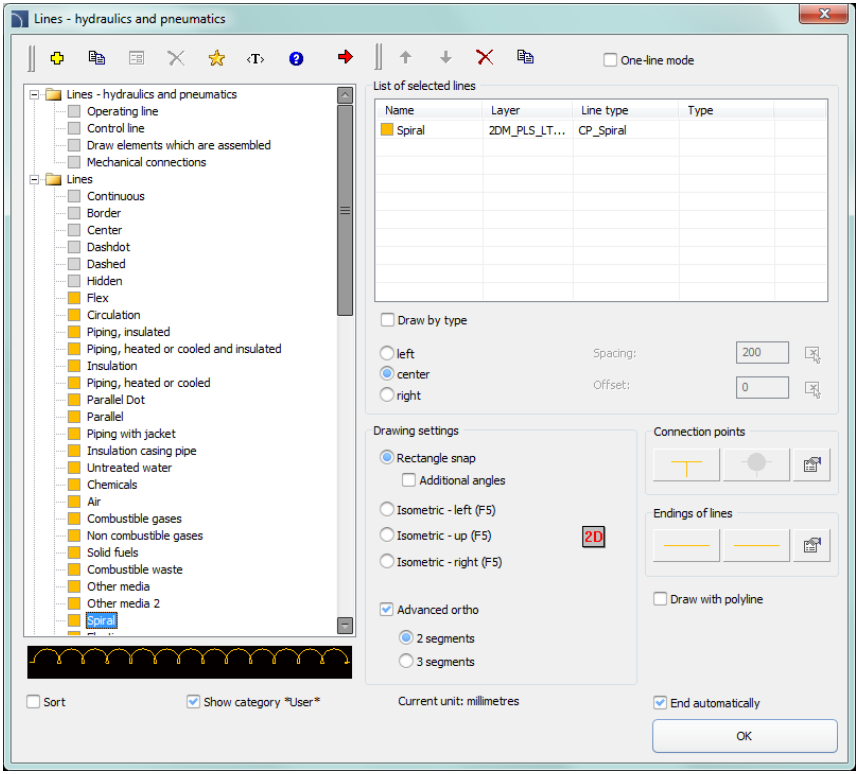
Creating pneumatic schemes

Hydraulics and pneumatics lines

The **Lines – hydraulics and pneumatics** command contains basic lines that are used in drawing of technological schemes, as well as, hydraulic and pneumatic schemes. It is possible to simultaneously draw many lines with a specified spacing and with a specified offset from objects that exist on the drawing. Program allows to define types of drawn lines, as well as to define user own system types.

In **CADprofi** program, schematical lines are mean as any pipes (pipe lines) or cables (electrical lines). Because of that all possibilities and options regarding the lines have been described in the **CADprofi HVAC & Piping** and **CADprofi Electrical** modules appropriate chapters (check page 148 and 222).

This command also contains general and special lines, e.g. terrain identification, concrete flooring description, boundary lines and others.



Lines – hydraulics and pneumatics dialog window

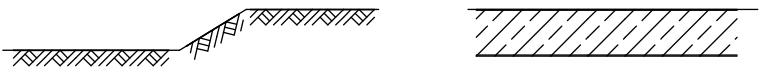
Procedures

Drawing lines

1. Run the **Lines – hydraulics and pneumatics** command.
2. In the dialog window select the desired line.
3. Click the **Add element to a list** button, or double click on it to put it on the **List of selected lines**.
4. (Optional) Repeat steps 2 and 3 in order to add more lines to the list.
5. (Optional) Using the **Copy selected rows**, **Remove selected rows** buttons add or remove the selected line from the list.
6. (Optional) Using the **Move selected rows down**/**Move selected rows up** buttons change the order of the lines on the list.
7. (Optional) Fill the details about line type, turn on/off the **Draw by type option**.
8. (Optional) If there is more than one line on the **List of selected lines** set the **Offset** value.
9. Specify the draw edge and the **Spacing value** if needed.
10. (Optional) If on the **List of selected lines** there is only one object you can set the **connection** and **line endings** options.
11. Enable/disable the **End automatically** option.
12. Click the **OK** button in order to start drawing a line by indicating points on the drawing. In order to finish click the **Enter** or **Esc** key.

Schematic lines are simple lines or polylines that's why it is also possible to draw lines with the CAD program standard commands such as **Line** or **Polyline**. However, it should be noted, that lines drawn in this way should be placed on appropriate layers. In order to match properties (e.g. layers), between the drawn objects, the **Match properties** command located in the CAD program can be used.

> ***Hint:***
>
> ***To edit lines the „Schematic lines –edit" command (check page 35) is used.***



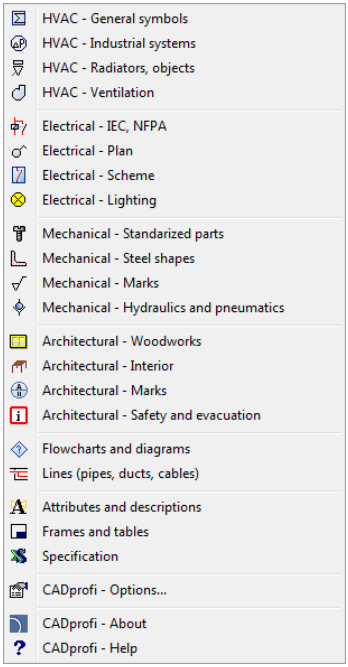
Example of different special lines with appropriate density scale (_LTSCALE)

CP–Symbols

CP–Symbols

Introduction

Program has got a basic library of symbols that are used to create projects from various branches. Four **CP–Symbols** series are available: Mechanical, Architectural, HVAC & Piping and Electrical. Each series consists of several libraries which are licensed separately or in packages. All program commands are available in a separate menu and in the **CP–Symbols** toolbar:

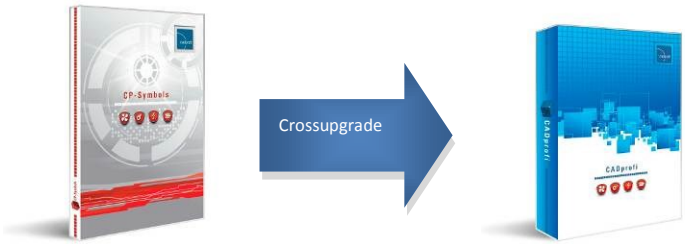


A more extensive program is the **"Full CADprofi version"** which also contains the corresponding CP-Symbols module (for example **CADprofi Electrical** possesses many additional features and libraries as well as it contains the library of symbols from **CP–Symbols Electrical**).

The installation and license activation process has been described in the **Installation** (check page 15) chapter.

Hint

When installing CP-Symbols, a demo version of CADprofi is also installed. Remember! For the difference in price between these programs you can buy a crossupgrade from CP-Symbols to "full CADprofi version". Ask your dealer about the crossupgrade or visit our website: www.cadprofi.com.



CP-Symbols HVAC & Piping Series

This series includes a basic library of symbols and objects used in sanitary and industry installation projects. This program is designed for people who only occasionally prepare industry drawings - mainly diagrams.

CP-Symbols HVAC & Piping consists of the following libraries:

HVAC – General symbols

Library possesses the most important symbols used in heating, water, sewage, fire safety and other technological installations.

HVAC – Industrial systems

Library possesses the most important symbols used in industrial installations schemes and P&ID schemes.

HVAC – Radiators, objects

Library contains sanitary equipment, radiators and other objects used in plans and strand diagrams.

HVAC – Ventilation

Library possesses a database of basic symbols used in ventilation and air condition installations by PN and DIN standards.

Flowcharts and diagrams

Library contains symbols that are used in block schemes in many branches.

Basic functionality of those commands is similar to the corresponding **CADprofi** program **Symbols** (check page 172) command.
More advanced features can be found only in the “full version” of **CADprofi HVAC & Piping**.

CP-Symbols Electrical Series

This series includes a basic library of symbols used in electrical installation projects. This program is designed for people who only occasionally prepare industry drawings - mainly diagrams.

CP-Symbols Electrical consists of the following libraries:

Electrical – IEC, NFPA

Library possesses basic symbols which were prepared in accordance with EN 60617 and NFPA 79 standards.

Electrical – Plan

Library possesses basic symbols of electrical installations that are used on architectural plans.

Electrical – Scheme

Library contains symbols used on a variety of schemes, including current installations, low voltage installations, telecommunication and other.

Electrical – Lighting

Library possesses a basic database of lighting fixtures.

Flowcharts and diagrams

Library contains symbols that are used in block schemes in many branches.

Basic functionality of those commands is similar to the corresponding **CADprofi** program **IEC, NFPA symbols** (check page 243) command.
More advanced features can be found only in the “full version” of **CADprofi Electrical**.

CP-Symbols Architectural Series

This series includes a basic library of symbols and objects used in architectural projects. This program is designed for people who only occasionally prepare building plans.

CP-Symbols Architectural consists of the following libraries:

Architectural – Woodworks

Library possesses the most important building elements (windows and doors) used in designing plans and elevations.

Architectural – Interior

Library possesses basic elements of interior furnishings, including: furniture, utensils, home appliances, culverts and other.

Architectural – Marks

Library possesses the most important marks used in architectural drawings, including: rooms marks, woodwork marks etc.

Architectural – Safety and evacuation

Library possesses symbols used on evacuation plans and Health and Safety symbols.

Flowcharts and diagrams

Library contains symbols that are used in block schemes in many branches.

Basic functionality of those commands is similar to the corresponding **CADprofi** program **Doors, windows** (check page 116) command.
More advanced features can be found only in the “full version” of **CADprofi Architectural**.

CP-Symbols Mechanical Series

This series includes a basic library fo symbols and standarized elements that are used in mechanical projects. This program is designed for people who only occasionally prepare technical drawings.

CP-Symbols Mechanical consists of the following libraries:

Mechanical – Standarized parts

Library possesses the most important elements from national and international standards, including: bolts, nuts, keys, wedges, bearings and other.

Mechanical – Steel shapes

Library possesses the most important shape profiles that are used in constructions, including: shapes, C-shapes, T-shapes and more.

Mechanical –Marks

Library possesses basic marks that are used in mechanical industry drawings.

Mechanical –Hydraulics and pneumatics

Library possesses basic symbols used in drawings of the hydraulic and pneumatic system schemes.

Flowcharts and diagrams

Library contains symbols that are used in block schemes in many branches.

Basic functionality of those commands is similar to the corresponding **CADprofi** program **Hydraulics and pneumatics symbols** (check page 324) command.
More advanced features can be found only in the “full version” of **CADprofi Mechanical**.

Other commands

CP-Symbols program also possesses general commands. Their description and usage is described in **CADprofi** chapters:

Attributes nad descriptions

Command used to edit attribute values of one or many **CADprofi** symbols.
Basic functionality of this command is similar to the corresponding **CADprofi** program **Attributes and descriptions** (check page 34) command.

Bill of materials

Command is used to create BOM from elements used in the drawing.
Basic functionality of this command is similar to the corresponding **CADprofi** program **Bill of materials** (check page 46) command.

Frames and tables

Command possesses a basic library of frames and title tables.
Basic functionality of this command is similar to the corresponding **CADprofi** program **Frames and tables** (check page 37) command.

Lines (pipes, ducts, cables)

This function allows to draw single-line lines in a schematic view.
Basic functionality of this command is similar to the corresponding **CADprofi** program e.g. **Pipes/Ducts – scheme** (check page 148) command.

> ***Notice***
>
> ***Some features of the above commands are only available in the CADprofi program.***

About and Options

The **CADprofi - About** (check page 27) and **CADprofi Options** (check page 28) commands in the **CP-Symbols** series are corresponding to the **full CADprofi program**.

CP–Manufacturers

CP–Manufacturers

Introduction

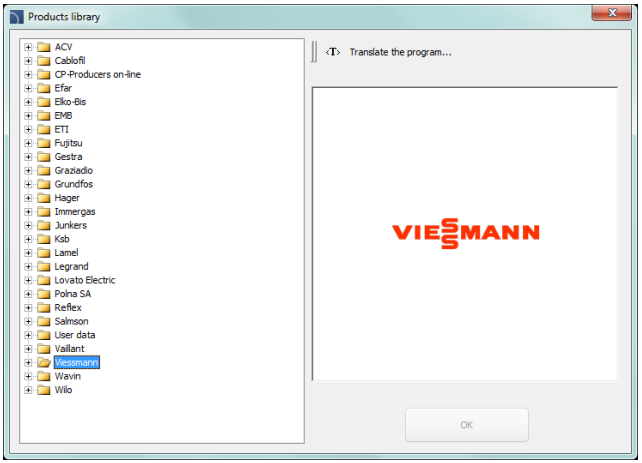
CP-Manufacturers program includes CAD libraries of many manufacturers of construction installation equipment and devices. Objects are presented in different views. Many objects are also available as 3D models. Thanks to the parametrical technology each individual libraries take up very little space on the disk. This technology also allows to automate many actions that are involved in designing and obtaining BOMs.

Libraries are also distributed on their own by manufacturers who have the license for **CP-Manufacturers** program.

Program commands

All program commands are available in the menu and in the **CP-Manufacturers** toolbar**:**

The **Products library** command allows to select the active library. After insertion of elements from a given library it is possible to open the dialog window of the same library by pressing the **Previous library**. Thanks to this the user omits the manufacturer selection window.



Product library selection window

After selecting the library the dialog window appears, in which we can view the devices or schemes of selected manufacturer. The library structure can be different for each manufacturer, nevertheless the way of handling the objects is similar to work, with standard **CADprofi** objects that are described in the branch modules.

Exemplary libraries:

- **Parametrical armature** (check page 202) – e.g. Efar, EMB, KSB, Polna, Wilo.
- **Parametrical objects** (check page 207)**, e.g. vessels, boilers etc.** – e.g. Viessmann, Vaillant, ACV.
- **Sewerage** (check page 201) – e.g. Wavin.
- **Modular units** (check page 267) – e.g. ETI, Legrand, Lamel.
- **Mesh trays** (check page 283) – e.g. Cablofil.
- **Busbars** (check page 288) – e.g. Legrand, PiN Energia.
- **Schemes** (check page 181) – e.g. AVC, Immergas, Vaillant, Viessmann.

Other commands

CP-Manufacturers program also possesses general commands:

- **Create alternative view** (check page 64).
- **Cablofil – Couple cable trays** (check page 283).
- **Cablofil – Match properties** (check page 285).
- **Attributes and descriptions** (check page 34).
- **Frames and tables** (check page 37).
- **Bill of materials** (check page 46).
- **CADprofi Options** (check page 28).
- **CADprofi – CADprofi - About** (check page 27).

Information for manufacturers

One of the most important manufacturers marketing strategies is to prepare and share technical information about products that are used during the design phase. Currently, the design phase is done on a computer, therefore it is worth to prepare libraries of products that can be easily used in CAD programs.

Many manufacturers have chosen the **CP-Manufacturers** program, which is ideal for the preparation of products libraries and free distribution, i.e. through the internet.

CP-Manufacturers program acts as an electronic catalog CAD library that helps and speeds up the designing phase. Program is free for users and works as an add-on for most popular CAD programs, such as AutoCAD, BricsCAD, GstarCAD, progeCAD, ZWCAD, IntelliCAD and others. **CP-Manufacturers** program is currently used by tens of thousands of users in many countries. Data that is prepared for the **CP-Manufacturers** program is also included in the commercial version of **CADprofi** application, thanks to this it can be exported e.g. to the online CAD catalogs.

How to prepare data for CP-Manufacturers program

CAD blocks can be prepared and added to the program on their own by each entity. It is also possible to cooperate with the manufacturer of **CADprofi** program and with the program distributors. In each of these cases we provide an "open" version of **CADprofi** program, which allows to automatically add drawings to the library, create descriptions, add technical data to the objects, photographs or catalog numbers. Documentation and training or consultation regarding the library editing is also provided.

CP-Manufacturers program functionality:

- Possibility to work in all popular CAD programs, i.e. AutoCAD, BricsCAD, GstarCAD, IntelliCAD, progeCAD, ZWCAD and others. After installation, the configuration program is available, which allows to "attach" the add-on the found CAD systems. At any time, it is possible to "detach" the add-on from the CAD program, thanks to which user restore the original settings of his currently used environment.
- Configuration program allows to browse libraries, technical data and catalog numbers even if user hasn't got a CAD program installed.
- In the program, it is possible to use the **parametrical** technology, in which one object can represent multiple objects.
- Objects can have different properties that automate the work. An example is the automatic joining of objects in the one-click technology. Program validates the project correctness and warns the user if the joined objects don't match.

- For all objects it is possible to add technical data, photographs, catalog cards, descriptions, calculation programs or spreadsheets, e.g. for the equipment selection.
- The possibility to work in both 2D and 3D. In projects it is possible to combine 2D and 3D technologies with the function of automatic transition from 2D to 3D views, and vice versa.
- The possibility to add to the library technological schemes (e.g. boiler schemes, control systems, electrical switchboards etc.). Schemes can be easily adapted to the specific projects.
- Drawings that are created with the program can be edited in any CAD system without the need to install the add-on. It is important for the cooperation between many designers that work on one project (not every designer must have the add-on installed).
- Automatic description of objects and a convenient edition of attributes of all objects.
- An on-line update system.
- The possibility to introduce a system of automatic user registration.
- A BOM creator that allows to acquire data about objects that are used in the project.
- The possibility to choose the design unit (mm, cm, m or inches).
- System of automatic layers management which increases the drawings visibility. Appropriate printing styles are also included in the program.
- Possibility to work in multiple languages. Currently the interface is translated in 22 languages.
- Product libraries can also be added to the **CADprofi** program, thanks to which the prepared data will be available to many designers.
- Parametrical libraries can be exported to dwg, dxf, step files and other.
- Program possesses a complete system of on-line help. It is also possible to add presentations or tutorial videos.